

NASA/TM-1998-208718



Progressive Damage Analysis of Laminated Composite (PDALC) - (A Computational Model Implemented in the NASA COMET Finite Element Code) Version 2.0

*Timothy W. Coats and Charles E. Harris
Langley Research Center, Hampton, Virginia*

*David C. Lo
Sandia National Laboratories, Albuquerque, New Mexico*

*David H. Allen
Texas A&M University, College Station, Texas*

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

October 1998

Available from the following:

NASA Center for AeroSpace Information (CASI)
7121 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 487-4650

Abstract

A method for analysis of progressive failure in the Computational Structural Mechanics Testbed is presented in this report. The relationship employed in this analysis describes the matrix crack damage and fiber fracture via kinematics-based volume-averaged damage variables. Damage accumulation during monotonic and cyclic loads is predicted by damage evolution laws for tensile load conditions. The implementation of this damage model required the development of two testbed processors. While this report concentrates on the theory and usage of these processors, a complete listing of all testbed processors and inputs that are required for this analysis are included. Sample calculations for laminates subjected to monotonic and cyclic loads were performed to illustrate the damage accumulation, stress redistribution, and changes to the global response that occurs during the loading history. Residual strength predictions made with this information compared favorably with experimental measurements.

Introduction

Laminated composite structures are susceptible to the development of microcracks during their operational lives. While these microcracks tend to aggregate in high stress regions and result in localized regions of reduced stiffness and strength, the microcracks can affect the global response of the structure. This change in the global structure in turn can create high stresses and increase damage accumulation in another part of the structure. Thus to accurately predict the structural response and residual strength of a laminated composite structure, the effects of the accumulating damage must be incorporated into the global analysis. The approach taken is to develop damage-dependent constitutive equations at the ply level. These equations are then employed in the development of the lamination equations from which the constitutive module of the structural analysis algorithm is constructed. This algorithm is executed in a stepwise manner in which the damage-dependent ply-level results are used in the calculation of the global response for the next load step. This report will describe two Computational Structural Mechanics (CSM) Testbed (COMET) processors that were developed for the performance of such an analysis. A brief review of the theory behind the processors is first presented. The usage of these processors is then demonstrated. Since this analysis requires the use of other COMET processors, this report serves as a supplement to the *Computational Structural Mechanics Testbed User's Manual* (ref. 1).

It should be noted that the current damage model capability, computer code version 1.1, is limited to matrix cracking and fiber fracture under tensile loads.

Symbols and Abbreviations

A laminate extensional stiffness matrix
B laminate coupling stiffness matrix

β mode I matrix cracking scale factor
D laminate bending stiffness matrix
dpara material parameter determined from experimental data
 E_{11} lamina longitudinal modulus
 E_{22} lamina transverse modulus
F applied force
 G_{12} lamina shear modulus
 γ tensile fiber fracture scale factor
 \tilde{k} material parameter determined from experimental data
 N_x applied load
 \tilde{n} material parameter determined from experimental data
PDALC Progressive Damage Analysis of Laminated Composites
R percent of maximum load
 σ_{11}^c lamina longitudinal critical stress
 σ_{12}^c lamina shear critical stress
 σ_{22}^c lamina transverse critical stress
 Ψ mode II matrix cracking scale factor
 t_{ply} ply thickness
u longitudinal extension
 u^o, v^o, w^o undamaged midplane displacements
x, y, z displacement fields
 ν_{12} lamina Poisson's ratio

Damage-Dependent Constitutive Relationship

The damage-dependent constitutive relationship employed in the COMET analysis is based on a continuum damage mechanics model proposed by Allen, Harris, and Groves (refs. 2 and 3). Rather than explicitly modeling each matrix crack in the material, the averaged kinematic effects of the matrix cracks in a representative volume are modeled by internal state variables. These internal state variables for matrix cracks are defined by the volume-averaged dyadic product of the crack face displacement, u_i , and the crack face normal, n_j , as proposed by Vakulenko and Kachanov (ref. 4):

$$\alpha_{Lij}^M = \frac{1}{V_L} \int_S u_i n_j ds, \quad (1)$$

where α_{Lij}^M is the second order tensor of internal state variables, V_L is the local representative volume in the deformed state, and S is the crack surface area. This product can be interpreted as additional strains incurred by the material as a result of the internal damage. From micromechanics it has been found that the effects of the matrix cracks can be introduced into the ply-level constitutive equation as follows (ref. 5):

$$\sigma_L = [Q]\{\epsilon_L - \alpha_L^M\}, \quad (2)$$

where σ_L are the locally averaged components of stress, $[Q]$ is the stiffness matrix in ply coordinates, and $\{\epsilon_L\}$ are the locally averaged components of strain. The laminate constitutive relationships are obtained by integrating the ply constitutive equations through the thickness of the laminate to produce

$$\{N\} = [A]\{\epsilon_L^0\} + [B]\{\kappa_L\} + \{f^M\}, \quad (3)$$

$$\{M\} = [B]\{\epsilon_L^0\} + [D]\{\kappa_L\} + \{g^M\}, \quad (4)$$

where $\{N\}$ and $\{M\}$ are the resultant force and moment vectors, respectively; $[A]$, $[B]$, and $[D]$ are the well known laminate extensional, coupling, and bending stiffness matrices, respectively (ref. 6);

$\{\epsilon_L^0\}$ is the midplane strain vector; and $\{\kappa_L\}$ is the midplane curvature vector; $\{f^M\}$ and $\{g^M\}$ are the damage resultant force and moment vectors for matrix cracking, respectively (ref. 7). The application of $\{f^M\}$ and $\{g^M\}$ to the undamaged

material will produce midplane strain and curvature contributions equivalent to those resulting from the damage-induced compliance increase.

As the matrix cracks accumulate in the composite, the corresponding internal state variables must evolve to reflect the new damage state. The rate of change of these internal state variables is governed by the damage evolutionary relationships. The damage state at any point in the loading history is thus determined by integrating the damage evolutionary laws. Based on the observation that the accumulation of matrix cracks during cyclic loading is related to the strain energy release rate G in a power law manner (ref. 8). Lo et al. (ref. 9) have proposed the following evolutionary relationship for the internal state variable corresponding to mode I (opening mode) matrix cracks:

$$d\alpha_{L22}^M = \frac{d\alpha_{L22}^M}{dS} \bar{k} G^{\bar{n}} dN, \quad (5)$$

The term $d\alpha_{L22}^M$ reflects the changes in the internal state variable with respect to changes in the crack surfaces. This term can be calculated analytically from a relationship that describes the average crack surface displacements in the pure opening mode

(mode I) for a medium containing alternating 0°

and 90° plies (ref. 5). The term G is the strain energy release rate calculated from the ply-level damage-dependent stresses. The material

parameters, \bar{k} and \bar{n} , are phenomenological in nature and must be determined from experimental data (refs. 10 and 11). Because \bar{k} and \bar{n} are assumed to be material parameters, the values determined from one laminate stacking sequence should be valid for other laminates as well. Since the interactions with the adjacent plies and damage sites are implicitly reflected in the calculation of the ply-level response through the laminate averaging process, equation 5 is not restricted to any particular laminate stacking sequence.

When the material is subjected to quasi-static (monotonic) loads, the incremental change of the internal state variable is assumed to be

$$d\alpha_{kl}^M = \begin{cases} f(\epsilon_{kl}, \beta, \gamma, \psi) & \text{if } \epsilon_{kl} > \epsilon_{klcr}; \\ 0 & \text{if } \epsilon_{kl} < \epsilon_{klcr} \end{cases} \quad (6)$$

where ϵ_{klcrit} is the critical tensile failure strain and β, γ , and ψ are scale factors that describe the load carrying capability of the material after the occurrence of mode I (opening mode) matrix cracking, fiber fracture, and mode II (shear mode) matrix cracking, respectively. The physical interpretation of equation (2) is as follows: as long as the strains in a material element (local volume element or finite element) are less than the critical strains, ϵ_{klcrit} , no damage exists and the internal state variables have a zero value. When the strains reach their critical value, the element is damaged and this damage is represented by an internal state variable whose value is proportional to the local strain. The proportionality is dependent on the scale factors β, γ , and ψ . Currently, the scale factors are chosen such that when fiber fracture, mode II matrix cracking, or mode I matrix cracking occur in a ply within an element, the longitudinal, shear, and transverse stresses for that ply in that element are

$$\sigma_{11} = \gamma \sigma_{11}^{cr} \quad (7)$$

$$\sigma_{12} = \psi \sigma_{12}^{cr} \quad (8)$$

$$\sigma_{22} = \beta \sigma_{22}^{cr}, \quad (9)$$

where σ_{11}^{cr} , σ_{12}^{cr} , and σ_{22}^{cr} are the lamina longitudinal, shear, and transverse critical stresses, respectively.

Structural Analysis Formulation

In order to simplify the formulation, it is expedient to consider the special case of symmetric laminates. With this assumption, the coupling stiffness matrix, [B], becomes the null matrix and the in-plane and out-of-plane laminate equations are decoupled. The laminate equations (3) and (4) are then substituted into the plate equilibrium equations to yield the following governing differential equations for the plate deformations:

$$\begin{aligned} -p_x = & A_{11} \frac{\partial^2 u^o}{\partial x^2} + 2A_{16} \frac{\partial^2 u^o}{\partial x \partial y} + A_{66} \frac{\partial^2 u^o}{\partial y^2} \\ & + A_{16} \frac{\partial^2 v^o}{\partial x^2} + (A_{12} + A_{66}) \frac{\partial^2 v^o}{\partial x \partial y} \\ & + A_{26} \frac{\partial^2 v^o}{\partial y^2} + \frac{\partial f_1^M}{\partial x} + \frac{\partial f_2^M}{\partial y} \end{aligned} \quad (10)$$

$$\begin{aligned} -p_y = & A_{16} \frac{\partial^2 u^o}{\partial x^2} + (A_{12} + A_{66}) \frac{\partial^2 u^o}{\partial x \partial y} + A_{26} \frac{\partial^2 u^o}{\partial y^2} \\ & + A_{22} \frac{\partial^2 v^o}{\partial y^2} + 2A_{26} \frac{\partial^2 v^o}{\partial x \partial y} + A_{66} \frac{\partial^2 v^o}{\partial x^2} \\ & + \frac{\partial f_3^M}{\partial x} + \frac{\partial f_2^M}{\partial y} \end{aligned} \quad (11)$$

$$\begin{aligned} -p_z = & D_{11} \frac{\partial^4 w^o}{\partial x^4} + 4D_{16} \frac{\partial^4 w^o}{\partial x^3 \partial y} \\ & + 2(D_{12} + 2D_{66}) \frac{\partial^4 w^o}{\partial x^2 \partial y^2} + 4D_{26} \frac{\partial^4 w^o}{\partial x \partial y^3} \\ & + D_{22} \frac{\partial^4 w^o}{\partial y^4} - \frac{\partial^2 g_1^M}{\partial x^2} - \frac{\partial^2 g_2^M}{\partial y^2} - 2 \frac{\partial^2 g_3^M}{\partial x \partial y} \end{aligned} \quad (12)$$

These governing differential equations are integrated against variations in the displacement components to produce a weak form of the damage dependent laminated plate equilibrium equations. By substituting the corresponding displacement interpolation functions into the weak form of the plate equilibrium equations, the following equilibrium equations in matrix form are produced (ref. 12 and 13)

$$[K] \{\delta\} = \{F_A\} + \{F_M\} \quad (13)$$

where [K] is the element stiffness matrix, $\{\delta\}$ is the displacement vector, $\{F_A\}$ is the applied force vector, and $\{F_M\}$ is the damage-induced force vector resulting from matrix cracking. Note that the effects of the internal damage now appear on the right hand side of the equilibrium equations as damage-induced force vectors.

Structural Analysis Scheme

The progression of damage is predicted by an iterative and incremental procedure outlined in the flow chart shown in figure 1. The first block of figure 1 is a description of the finite element model. Block numbers are shown in the right hand corner of each box in the flowchart. Blocks 2 and 3 are processors that calculate the element stiffness matrices and assemble and factor the global stiffness matrix. The compliance changes due to damage are accounted for by combining the damage induced force vector with the applied force vector and solving for the global displacements in equation (6). This solution process occurs in block 4 and 5 and then the element stress resultants are computed in block 6.

The ply-level strains and stresses are computed in block 7 and as long as the strains in a material element (local volume element or finite element) are less than the critical strains, ϵ_{klcrit} , no damage exists and the internal state variables have zero values. When the strains reach their critical value, the element is damaged and this damage is represented by an internal state variable. The damage state is updated and the ply-level stresses and strains are post-processed in block 8. The analysis iterates over blocks 4 through 8 until equilibrium is established, and then the next load step is applied.

The implementation of this analysis into the COMET code can be accomplished with the development of processors DRF and DGI. These processors, as with other COMET processors, are semi-independent computational modules that perform a specific set of tasks. Processor DRF first calculates the damage resultant forces and moments and then incorporates them into the global force vectors. The second processor, DGI, post-processes the elemental stress resultants into ply level stresses and strains using the damage dependent constitutive relationship. With this information, the processor computes the damage evolution and updates the damage state for the next series of calculations. The remaining calculations can be performed with existing COMET processors. The following is a listing in order of COMET processor executions for this analysis:

1. Procedure ES defines element parameters.
2. Processor TAB defines joint locations, constraints, reference frames.
3. Processor AUS builds tables of material and section properties and applied forces.
4. Processor LAU forms constitutive matrix.
5. Processor ELD defines elements.
6. Processor E initializes element datasets, creates element datasets.
7. Procedure ES initializes element matrices.
8. Procedure ES calculates element intrinsic stiffness matrices.
9. Processor RSEQ resequences nodes for minimum total execution time.
10. Processor TOPO forms maps to guide assembly and factorization of system matrices.
11. Processor K assembles system stiffness matrix.
12. Processor INV factors system stiffness matrix.
13. Continue
14. Processor DRF forms damage resultant force vectors.
15. Processor SSOL solves for static displacements
16. Procedure STRESS calculates element stress resultants.
17. Processor DGI calculates ply level stresses and damage evolution.

18. For next load cycle, go to step 13; else stop.

The usage and theory behind each of the existing processors can be found in *The Computational Structural Mechanics Testbed User's Manual* (ref. 1). The processors DRF and DGI are described in Appendices A and B of this report, respectively. With the exception of processor DRF and DGI, other processors from the COMET processor library can be substituted into the listing above to perform the tasks specified.

Example Calculations

Example calculations were conducted using COMET to illustrate the features of the progressive damage code. The first example demonstrates the effects of the evolving matrix damage on a crossply laminated composite plate subjected to constant amplitude fatigue loads. The dimensions and boundary conditions for the laminated plate are shown in figure 2. This plate was discretized into 24 four-node quadrilateral EX47 shell elements (ref. 14). In this example, the plate has a $[0/90]_s$ laminate stacking sequence and the ply-level mechanical properties are listed in table 1. These properties corresponded to those measured for IM7/5260 (ref. 11). A maximum load of 2500 lb/in at an R-ratio of 0.1 was applied to the laminate. The COMET runstream and input, as well as a segment of the output, for this example can be found in the section entitled "Progressive Failure Analysis Input" of Appendix B.

The predicted distribution of the mode I matrix crack damage α_{22}^M in the 90° plies is shown in figure 3. The damage was greatest at the narrow end of the plate since the component of stress normal to the fiber was highest in this region. The higher stresses further translated to a greater amount of energy available for the initiation and propagation of additional damage. This was reflected in the damage evolution along the length of the plate. However as damage accumulated in the plate, the stress gradient in the 90° plies became less steep as shown in figure 4. The similarity in stress resulted in relatively uniform changes to the damage state at the higher load cycles. For this laminate stacking sequence, the load shed by the damaged 90° plies was absorbed by the 0° plies. The consequence of this load redistribution is an increase in the global displacements as illustrated in figure 5. The redistribution of load to the adjacent plies will affect the interlaminar shear stresses as well. This could create favorable conditions for the propagation of delamination.

The second example examines the effects of damage accumulation during cyclic fatigue loads on the stiffness of notched laminates. In this example, the notched laminates are tension fatigue loaded for 100,000 cycles. The notched (central circular hole) laminate is shown in figure 6. Symmetry was assumed about the length and width of the laminate so that only a quarter of the laminate was modeled by the finite element model. This model, also shown in figure 6, consisted of 320 four-node quadrilateral EX47 shell elements. Two laminate stacking sequences, a cross-ply $[0/90_s]$ and a quasi-isotropic $[0/\pm 45/90]$, were considered. These laminates possessed the same ply-level material properties as the first example. (See table 1.) The maximum fatigue loads employed in sample calculations are listed in table 2. The COMET runstream for the fatigue loading is similar to the previous example which is listed in Appendix B. In figure 7, the predicted stiffness loss for the open-hole geometry is compared to experimentally measured values of stiffness loss measured over a 4-in. gage length, symmetric about the open hole.

The final example considers the effects of monotonically increasing loads on the residual strength of AS4/938, AS4/8553-40, and AS4/3501-6 notched laminates (ref. 15). The notched laminate shown in figure 8 is considered in this example. Symmetry was assumed about the length and width of the laminate so that only a quarter of the laminate was modeled by the finite element model. The four inch wide panels with one inch notches had a mesh (figure 8) that consisted of 278 four node quadrilateral EX47 shell elements. All of the elements on the $x=0$ centerline from the notch-tip to the edge of the panel were the same size for the 4, 12, and 36 inch wide panels. The laminate stacking sequence was orthotropic $[\mp 45/0/90/\mp 30/0]$. These laminates possessed the ply-level material properties shown in table 3 (ref. 16 and 17). The applied load is incrementally increased with each load step to simulate a ramp up load input. Failure of the component is assumed to have occurred when the elements that span the width of the laminate have sustained a level of fiber fracture such that the analysis cannot reach equilibrium within a given load step. The load at which this condition exists is used to calculate the residual strength. The COMET runstream for this example is listed in Appendix C. The predicted residual strengths are shown along with experimental measurements in figure 9.

The load redistribution due to damage progression is simulated using monotonic damage growth parameters (β , ψ , and γ). These parameters

can be complicated algebraic functions describing the complex behavior of load redistribution due to matrix cracking and fiber fracture. However, for the purpose of developing the framework for this progressive damage methodology, simple constants were chosen for the parameters. So for this analysis, the damage growth law parameters govern the load redistribution in a way that is similar to the ply discount method. It is not reasonable to assume a 100% load redistribution at the instant of failure for mode I matrix cracking and tensile fiber fracture. Therefore, a 90% load redistribution was assumed, i.e. the local ply stress is only 10% of the critical ply strength ($\beta = 0.1$ and $\gamma = 0.1$). As the applied load increases, mode I matrix cracking and tensile fiber fracture internal state variables increase in proportion to the local strains. This results in a constant stress level (10% of the critical ply strength) in the damaged plies, illustrated in figure 10, with the load redistributing to the surrounding plies and elements.

Based on Iosipescu shear data (ref. 18) there is a shear strain (γ_{12}^0) where the behavior is no longer linear and becomes almost perfectly plastic. At this strain level, ψ is equal to 1.0 to simulate elastic-perfectly plastic shear stress/strain behavior as illustrated in figure 10. This implies that as the applied load increases, the damaged ply carries 100% of the critical shear strength while the additional stress transfers to the surrounding plies and elements. When the shear strain becomes catastrophic (γ_{12}^{cr}), ψ is assumed to be equal to 0.1. The load redistribution for shear is now similar to the case for mode I matrix cracking.

Concluding Remarks

This report describes a progressive failure analysis for laminated composites that can be performed using the Computational Structural Mechanics (CSM) Testbed (COMET) finite element code. The present analysis utilizes a constitutive model that describes the kinematics of the matrix cracks via volume averaged internal state variables. The evolution of these internal state variables is governed by an experimentally based damage evolutionary relationship. The nonlinearity of the constitutive relationship and of the damage accumulation process requires that this analysis be performed incrementally and iteratively.

Two processors were developed to perform the necessary calculations associated with this constitutive model. In the analysis scheme, these processors were called upon to interact with existing COMET processors to perform the progressive failure analysis. This report, which serves as a guide for performing progressive failure analysis on

COMET, provides a brief background on the constitutive model and the analysis methodology in COMET. The description and usage of the two progressive failure processors can be found in the appendices of this report.

The results from the example problems illustrated the stress redistribution that occurred during the accumulation of matrix cracks and fiber fracture. This in turn influenced the damage evolution characteristics, the global displacements,

and the residual strengths. It should be noted that the current damage model capability is limited to mode I (opening mode) and mode II (shearing mode) matrix cracking and fiber fracture under tensile loading conditions. The inclusion of other damage modes such as delamination and compression failure mechanisms will provide a more complete picture of the failure process.

Appendix A

Processor DRF

A1. General Description

This processor calculates the damage resultant forces and moments caused by matrix cracking in laminated composites. These resultant forces and moments when applied to an undamaged laminate will produce an equivalent amount of displacements and curvatures to those resulting from the matrix crack surface kinematics in a damaged laminate. This enables an analysis of the response of a damaged laminate without having to update the stiffness matrix each time the damage state changes. Matrix crack damage is modeled in this processor by volume averaged crack surface kinematics using internal state variables (refs. 2 and 3).

Processor DRF and processor DGI, which is described in appendix B, were developed to perform progressive failure analysis of quasi-static and fatigue loaded laminates in Computational Structural Mechanics (CSM) Testbed (COMET). Analyses from these processors are stored in two formats. One is in standard format that is accessed by opening the output file. The other is a data set, which is stored in a testbed data library, and provides data to processors and post-processors (ref. 1). In this analysis, processor DRF is used in conjunction with COMET analysis processors to determine the static displacement and elemental stress resultants for a laminated composite structure containing matrix crack damage. Processor DGI then calculates the damage-dependent ply stresses. The damage state is updated based on the ply stresses and this procedure is repeated for the next load cycle.

A1.1. Damage Dependent Constitutive Relationship

In this processor, the effects of the matrix cracks are introduced into the ply-level constitutive equations as follows (ref. 3):

$$\sigma_L = [Q]\{\epsilon_L - \alpha_L^M\}, \quad (A1)$$

where σ_{Lij} are the locally averaged components of stress, $[Q]$ is the ply level reduced stiffness matrix, and ϵ_{Lij} are the locally averaged components of strain. α_{Lij}^M are the components of the strain-like internal state variable for matrix cracking and are defined by

$$\alpha_{Lij}^M = \frac{1}{V_L} \int_S u_i n_j ds, \quad (A2)$$

where V_L is the volume of an arbitrarily chosen representative volume of ply thickness which is sufficiently large that α_{Lij}^M do not depend on V_L , u_i are the crack opening displacements, n_j are the components of the vector normal to the crack face, and S is the surface value of the volume V_L . The present form of the model assumes that α_{11}^M , α_{22}^M , and α_{33}^M , the internal state variables representing the tensile fiber fracture, mode II (shear mode) matrix cracking, and mode I (opening mode) matrix cracking, respectively, are the only nonzero damage components.

A1.2. Damage Dependent Laminate Equations

The ply level strains are defined as follows:

$$\epsilon_{Lxx} = \epsilon_{Lxx}^o - z \kappa_{Lxx} \quad (A3)$$

$$\epsilon_{Lyy} = \epsilon_{Lyy}^o - z \kappa_{Lyy} \quad (A4)$$

$$\epsilon_{Lxy} = \epsilon_{Lxy}^o - \kappa_{Lxy} \quad (A5)$$

where ϵ_L^o and κ_L are the midplane strains and curvatures, respectively. The aforementioned ply strains are then substituted into equation (A1) to produce the ply-level stresses. Damage-dependent lamination equations are obtained by integrating these ply stresses through the thickness of the laminate (ref. 15). Next, the stiffness matrix in the laminate equation is inverted to produce

$$\begin{Bmatrix} \epsilon_L^o \\ \kappa_L \end{Bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix}^{-1} \begin{Bmatrix} N - f^M \\ M - g^M \end{Bmatrix} \quad (A6)$$

where [A], [B], and [D] are, respectively, the undamaged laminate extensional, coupling, and bending stiffness matrices. They are defined by the following equations from reference 6:

$$[A] = \sum_{k=1}^n [\bar{Q}]_k (z_k - z_{k-1}) \quad (A7)$$

$$[B] = \frac{1}{2} \sum_{k=1}^n [\bar{Q}]_k (z_k^2 - z_{k-1}^2) \quad (A8)$$

$$[D] = \frac{1}{3} \sum_{k=1}^n [\bar{Q}]_k (z_k^3 - z_{k-1}^3) \quad (A9)$$

where $[\bar{Q}]_k$ is the transformed reduced elastic modulus matrix for the k^{th} ply in laminate coordinates. In equation (A6), N are the components of the resultant force per unit length and M are the components of the resultant moments per unit length. The variables $\{f^M\}$ and $\{g^M\}$ represent the contribution to the resultant forces and moments from matrix cracking and are calculated from,

$$\{f^M\} = - \sum_{k=1}^n [\bar{Q}]_k (z_k - z_{k-1}) \{\alpha_L^M\}_k \quad (A10)$$

$$\{g^M\} = - \frac{1}{2} \sum_{k=1}^n [\bar{Q}]_k (z_k^2 - z_{k-1}^2) \{\alpha_L^M\}_k \quad (A11)$$

where $\{\alpha^M\}_k$ contains the matrix cracking internal state variables for the k^{th} ply. Thus given the forces N and moments M, as well as the damage variables in each ply, equation (A6) can be utilized to calculate the midsurface strains ϵ_L^o and curvature κ_L .

A2. Processor SYNTAX

This processor uses keywords and qualifiers along with the CLIP command syntax (ref. 1). Two keywords are recognized: SELECT and STOP.

A2.1 Keyword SELECT

This keyword uses the qualifiers listed below to control the processor execution.

| Qualifier | Default | Meaning |
|-----------|-----------|---|
| LIBRARY | 1 | Input and output library. |
| ELEMENT | ALL | Element type (EX47, EX97) used in the analysis. Default is all element types found in LIBRARY. |
| SREF | 1 | Stress reference frame. Stress resultants may have been computed in the element stress/strain reference frame (SREF=0) or in one of three alternate reference frames. For SREF=1, the stress/strain x-direction is coincident with the global y-direction. For SREF=3 the stress/strain x-direction is coincident with the global z-direction. <u>Note that the processor currently must have the stress/strain coincident with the global x-direction (SREF=1).</u> |
| PRINT | 1 | Print flag. May be 0, 1, or 2; 2 results in the most output. |
| MEMORY | 2 000 000 | Maximum number of words to be allocated in blank common. This is an artificial cap on memory put in place so that the dynamic memory manager does not attempt to use all of the space available on the machine in use. |
| DSTATUS | 1 | Damage state flag. If no damage, DSTATUS = 0. If matrix cracking (cyclic load), DSTATUS = 1. If matrix cracking (monotonic load), DSTATUS = 22222. |
| XFACTOR | 0.0 | Increases the specified applied forces by this factor at every load step. This is used in the residual strength calculations. |

A2.2 Keyword STOP

This keyword has no qualifiers.

A3. Subprocessors and Commands

Processor DRF does not have subprocessors

A4. Processor Data Interface

A4.1. Processor Input Datasets

Several datasets, listed below, are used as input for processor DRF.

| Input Dataset | Contents |
|---------------|---|
| ELTS.NAME | Element names |
| OMB.DATA.1.1 | Material properties including strain allowables |

| | |
|-----------------|---|
| LAM.OMB.*.* | Laminate stacking sequence |
| ES.SUMMARY | Various element information |
| PROP.BTAB.2.102 | ABD matrix |
| WALL.PROP.1.1 | Shell wall dataset |
| DIR.xxxx.*.* | Element directory dataset |
| DEF.xxxx.*.* | Element definition (connectivity) dataset |
| ISV.xxxx.*.* | Internal state variable dataset |
| xxxx.EFIL.*.* | Element nodal coordinates and transformations |
| APPL.FORC | Applied force and moments at joints |

A4.2 Processor Output Datasets

These Datasets used as output for processor DRF.

| Output Dataset | Contents |
|----------------|--|
| APPL.FORC | Applied force and moments at joints |
| DFCT.xxxx.*.* | Temporary damage resultant force dataset |
| DRFC.xxxx.*.* | Damage resultant force dataset |

A5. Limitations

Only EX47 and EX97 elements implemented with the generic element processor ES1 will be processed by processor DRF. All other elements will be ignored. The stress reference frame must be coincident with the global x -direction.

A6. Error Messages

Fatal errors will occur when any of the required datasets are missing from the input data library or when the stress resultants at the integration points are missing.

Warning messages will be written and execution will continue when there is a missing or unreadable keyword or qualifier or if any of the original SPAR elements are encountered.

A7. Usage Guidelines and Examples

A7.1 Runstream Organization

The following listing illustrates the organization of a progressive failure analysis that uses COMET. Because of the nonlinear nature of the damage-dependent constitutive equation, this analysis is performed in a stepwise manner.

- | | |
|------------------|---|
| 1. Procedure ES | Define element parameters. |
| 2. Processor TAB | Define joint locations, constraints, reference frames. |
| 3. Processor AUS | Build tables of material and section properties and applied forces. |
| 4. Processor LAU | Form constitutive matrix. |

- | | |
|--|---|
| 5. Processor ELD | Define elements. |
| 6. Processor E | Initialize element datasets, create element datasets. |
| 7. Procedure ES | Initialize element matrices. |
| 8. Procedure ES | Calculate element intrinsic stiffness matrices. |
| 9. Processor RSEQ | Resequence nodes for minimum total execution time. |
| 10. Processor TOPO | Form maps to guide assembly and factorization of system matrices. |
| 11. Processor K | Assemble system stiffness matrix. |
| 12. Processor INV | Factor system stiffness matrix. |
| 13. Continue | |
| 14. Processor DRF | Form damage resultant force vectors. |
| 15. Processor SSOL | Solve for static displacements |
| 16. Procedure STRESS | Calculate element stress resultants |
| 17. Processor DGI | Calculate ply level stresses and damage evolution |
| 18. For next load cycle, go to step 13; else stop. | |

A7.2. Progressive Failure Analysis Input and Output

Please refer to Processor DGI for usage example (Appendix 1.2).

A8. Structure of Datasets Unique to Processor DRF

A8.1. DRFC.xxxx

This data set is created by processor DRF and uses the SYSVEC format. See APPL.FORC.iset.1 This dataset contains the damage resultant forces and moments corresponding to the given matrix cracking state.

A8.2. DFCT.xxxx

Data set DFCT.xxx is created by processor DRF and uses the SYSVEC format. See APPL.FORC.iset.1 This dataset contains the damage resultant forces and moments from the previous load step and is used to restore the applied force vector to the initial value.

A8.3. ISV.xxxx

This data set contains the matrix cracking internal state variables at each layer. The xxxx is the element name. The data is stored in a record named ALPAM.1. This record contains n items, where

$$n = n_{layer} \times n_{intgpt} \times n_{elt}$$

and n_{layer} is the number of layers in the model, n_{intgpt} is the number of integration points for the element, and n_{elt} is the number of elements.

The data is stored in the following order:

1. $\alpha_{L_{11}}^M$, internal state variable associated with fiber fracture.
2. $\alpha_{L_{22}}^M$, internal state variable associated with mode I opening of the matrix crack.
3. $\alpha_{L_{12}}^M$, internal state variable associated with mode II opening of the matrix crack.

The data storage occurs for every layer, every integration point, and every element.

Appendix B

Processor DGI

B1. General Description

Processor DGI predicts the evolution of matrix crack damage in laminated composites for monotonical loads and cyclic fatigue loads. The processor also calculates fiber fracture under tensile load conditions. The matrix crack damage is represented in this processor by volume-averaged crack surface kinematics that use internal state variables (refs. 2 and 3). The evolution of these internal state variables is governed by a phenomenological growth law.

This processor was designed to perform progressive failure analysis of laminated composite structures in the Computational Structural Mechanics (CSM) Testbed (COMET). At each load cycle, the elemental stress resultants for a laminated composite structure are obtained from COMET with the effects of matrix crack damage accounted for by Processor DRF. Processor DGI then postprocesses this information and uses the ply-level stresses to determine the evolution of matrix crack damage in each ply of the laminate. This procedure is repeated until the specified number of load cycles has been reached.

B1.1 Damage Dependent Constitutive Relationship

In this processor, the effects of the matrix cracks are introduced into the ply-level constitutive equations as follows (ref. 5):

$$\{\sigma_L\} = [Q]\{\epsilon_L - \alpha_L^M\} \quad (B1)$$

where $\{\sigma_L\}$ are the locally averaged components of stress, $[Q]$ is the ply level reduced stiffness matrix, and $\{\epsilon_L\}$ are the locally averaged components of strain. $\{\alpha_L^M\}$ are the components of the strain-like internal state variable for matrix cracking and are defined by

$$\alpha_{L_{ij}}^M = \frac{1}{V_L} \int_S u_i n_j dS \quad (B2)$$

where V_L is the volume of an arbitrarily chosen representative volume of ply thickness that is sufficiently large that $\alpha_{L_{ij}}^M$ do not depend on V_L , u_i are the crack opening displacements, and n_j are the components of the vector normal to the crack face. The present form of the model assumes that $\alpha_{11}^M, \alpha_{12}^M$, and α_{22}^M , the internal state variables representing the tensile fiber fracture, mode II (shear mode) matrix cracking, and mode I (opening mode) matrix cracking, respectively, are the only nonzero damage components.

For a uniaxially loaded medium containing alternating 0° and 90° plies, $\alpha_{L_{22}}^M$ has been found from a micromechanics solution to be related to the far field normal force and crack spacing as follows (ref. 5):

$$\alpha_{L_{22}}^M = \frac{\frac{\rho}{2\bar{t}}}{\frac{\pi^4}{64\xi} - C_{2222}} \quad (B3)$$

where

$$\xi = \sum_{m=1}^{\infty} \sum_{n=1}^{\infty} \frac{1}{C_{2222}(2m-1)^2(2n-1)^2 + C_{1212}(\tilde{a}/\tilde{t})^2(2n-1)^4} \quad (B4)$$

ρ is the force per unit length that is applied normal to the fibers and $2\tilde{t}$ and $2\tilde{a}$ are the layer thickness and crack spacing, respectively. The C_{2222} is the modulus in the direction transverse to the fibers and C_{1212} is the in-plane shear modulus. Both moduli are the undamaged properties.

B1.2. Damage Evolution Relationship

Equation (B3) is used when the matrix crack spacing is known in each ply of the laminate. Since it is usually necessary to predict the damage accumulation and response for a given load history, damage evolutionary relationships must be utilized to determine the values of the internal state variables. The following relationship was used for the rate of change of the internal state variable α_{L22}^M in each ply during fatigue loading conditions (ref. 9):

$$d\alpha_{L22}^M = \frac{d\alpha_{L22}^M}{dS} \tilde{k} G^{\tilde{n}} dN \quad (B5)$$

where $d\alpha_{L22}^M$ describes the change in the internal state variable for a given change in the crack surface areas,

\tilde{k} and \tilde{n} are material parameters (refs. 10 and 11), N is the number of load cycles, and G is the damage-dependent strain energy release rate for the ply of interest and is calculated from the following equation:

$$G = V_L C_{ijkl} (\epsilon_{Lij} - \alpha_{Lij}^M) \frac{d\alpha_{Lij}^M}{dS} \quad (B6)$$

where V_L is the local volume. Interactions with the adjacent plies will result in ply strains ϵ_{Lij} , that are affected by the strains in adjacent plies. Thus, the strain energy release rate G in each ply will be implicitly reflected in the calculation of the ply-level response, so that equation (B5) is not restricted to a particular laminate stacking sequence. Substituting equation (B6) in equation (B5) and integrating the result in each ply over time gives the current damage state in each ply for any fatigue load history.

When the material is subjected to monotonically increasing loads, the rate of change of the internal state variable α_{Lij}^M is described by

$$d\alpha_{11} = \begin{cases} \epsilon_{11} - \alpha_{11}^{old} + \frac{1}{Q_{11}Q_{22} - Q_{12}^2} (Q_{12}\beta S_{cr}^y - Q_{22}\gamma S_{cr}^x) & \text{if } \alpha_{22} > 0; \\ \epsilon_{11} - \alpha_{11}^{old} + \frac{Q_{12}}{Q_{11}} \epsilon_{22} - \frac{\gamma S_{cr}^x}{Q_{11}} & \text{if } \alpha_{22} = 0 \end{cases} \quad (B7)$$

$$d\alpha_{22} = \begin{cases} \epsilon_{22} - \alpha_{22}^{old} + \frac{1}{Q_{11}Q_{22} - Q_{12}^2} (Q_{12}\gamma S_{cr}^x - Q_{11}\beta S_{cr}^y) & \text{if } \alpha_{11} > 0; \\ \epsilon_{22} - \alpha_{22}^{old} + \frac{Q_{12}}{Q_{22}} \epsilon_{11} - \frac{\beta S_{cr}^y}{Q_{22}} & \text{if } \alpha_{11} = 0 \end{cases} \quad (B8)$$

$$d\alpha_{12} = \frac{-\psi S_{cr}^{xy}}{Q_{66}} + \gamma_{12} - \alpha_{12}^{old} \quad (B9)$$

for strains exceeding $\epsilon_{11_{crit}}$, $\epsilon_{22_{crit}}$, and $\gamma_{12_{crit}}$, respectively. If none of the critical strains are exceeded, there is no damage. The α_{kl} is the updated internal state variable, α_{kl}^{old} is the internal state variable for the previous damage state, and $d\alpha_{11}$, $d\alpha_{22}$, and $d\alpha_{12}$ are the incremental changes in the internal state variables for tensile fiber fracture, mode I matrix cracking, and mode II matrix cracking, respectively. The monotonic damage growth parameters (β , ψ , and γ) can be complicated algebraic functions describing the complex behavior of load redistribution due to matrix cracking and fiber fracture. However, for the purpose of developing the framework for this progressive damage methodology, simple constants were chosen for the parameters. So for this analysis, the damage growth law parameters govern the load redistribution in a way that is similar to the ply discount method. It is not reasonable to assume a 100% load redistribution at the instant of failure for mode I matrix cracking and tensile fiber fracture. Therefore, a 90% load redistribution was assumed, i.e. the local ply stress is only 10% of the critical ply strength ($\beta = 0.1$ and $\gamma = 0.1$). As the applied load increases, mode I matrix cracking and tensile fiber fracture internal state variables increase in proportion to the local strains. This results in a constant stress level (10% of the critical ply strength) in the damaged plies, illustrated in figure 10, with the load redistributing to the surrounding plies and elements.

Based on Iosipescu shear data (ref. 16) there is a shear strain (γ_{12}^0) where the behavior is no longer linear and becomes almost perfectly plastic. At this strain level, ψ is equal to 1.0 to simulate elastic-perfectly plastic shear stress/strain behavior as illustrated in figure 10. This implies that as the applied load increases, the damaged ply carries 100% of the critical shear strength while the additional stress transfers to the surrounding plies and elements. When the shear strain becomes catastrophic (γ_{12}^{cr}), ψ is assumed to be equal to 0.1. The load redistribution for shear is now similar to the case for mode I matrix cracking.

B2. PROCESSOR SYNTAX

This processor uses keywords and qualifiers along with the CLIP command syntax. Two keywords are recognized: SELECT and STOP.

B2.1 Keyword SELECT

This keyword uses the qualifiers listed below to control the processor execution.

| Qualifier | Default | Meaning |
|-----------|---------|--|
| LIBRARY | 1 | Input and output library. |
| ELEMENT | ALL | Element type (EX47, EX97) used in the analysis. Default is all element types found in LIBRARY. |
| LOAD_SET | 1 | Load set; i of input data set STRS.xxxx. i . j . |
| SREF | 1 | Stress reference frame. Stress resultants may have been computed in the element stress/strain reference frame (SREF=0) or in one of three alternate reference frames. For SREF=1, the stress/strain x -direction is coincident with the global y -direction. For SREF=3 the stress/strain x -direction is coincident with the global z -direction. Note that the processor currently must have the stress/strain coincident with the global x -direction (SREF=1). |
| PRINT | 1 | Print flag. May be 0, 1, or 2; 2 results in the most output. |
| STEP | 0 | Step number in nonlinear analysis (i.e., i in the STRS.xxxx. i .0 data set for nonlinear analysis). |

| | | |
|---------|-----------|---|
| MEMORY | 2 000 000 | Maximum number of words to be allocated in blank common. This is an artificial cap on memory put in place so that the dynamic memory manager does not attempt to use all of the space available on the machine in use. |
| DSTATUS | 1 | Damage state flag. If no damage, DSTATUS = 0. If matrix cracking (cyclic load), DSTATUS = 1. If matrix cracking (monotonic load), DSTATUS = 22222. |
| INCSIZE | 1.0 | Increment size used in damage growth law. |
| NCYCLE | 1 | Cycle number |
| NINCR | 1 | Increment number |

B2.2 Keyword STOP

This keyword has no qualifiers.

B3. Subprocessors and Commands

None. Processor DGI does not have subprocessors.

B4. Processor Data Interface

B4.1 Processor Input Datasets

Several datasets, listed below, are used as input for processor DGI.

| Input Dataset | Contents |
|-----------------|---|
| ELTS.NAME | Element names |
| STRS.xxxx.i.j | Element stress resultants. Record named INTEG_PTS must exist. |
| OMB.DATA.1.1 | Material properties including strain allowables |
| LAM.OMB.*.* | Laminate stacking sequence |
| ES.SUMMARY | Various element information |
| PROP.BTAB.2.102 | ABD matrix |
| WALL.PROP.1.1 | Shell wall dataset |
| DIR.xxxx.*.* | Element directory dataset |
| DEF.xxxx.*.* | Element definition (connectivity) dataset |
| ISV.xxxx.*.* | Internal state variable dataset |
| DGP.DATA.1.1 | Damage growth law parameters data set |

B4.2 Processor Output Datasets

| Output Dataset | Contents |
|----------------|----------|
|----------------|----------|

| | |
|-------------|---|
| ISV.xxx.*.* | Internal state variable data set |
| PDAT.xxxx | Ply-level stresses, strains, and damage state |

B5. Limitations

Only EX47 and EX97 elements implemented with the generic element processor ES1 will be processed by processor DGI. All other elements will be ignored. The stress reference frame must be coincident with the global x -direction.

B6. Error Messages

Fatal errors will occur when any of the required datasets are missing from the input data library or when the stress resultants at the integration points are missing. (See section B4.1.)

Warning messages will be written and execution will continue when there is a missing or unreadable keyword or qualifier or if any of the original SPAR elements are encountered.

B7. Usage Guidelines and Examples

B7.1 Organization of Progressive Damage Analysis on Testbed

The organization of the COMET processors for a progressive failure analysis is shown below. The nonlinear nature of the damage-dependent constitutive equation requires that this analysis be performed in a stepwise manner.

- | | |
|--|---|
| 1. Procedure ES | Define element parameters. |
| 2. Processor TAB | Define joint locations, constraints, reference frames. |
| 3. Processor AUS | Build tables of material and section properties and applied forces. |
| 4. Processor LAU | Form constitutive matrix. |
| 5. Processor ELD | Define elements. |
| 6. Processor E | Initialize element datasets, create element datasets. |
| 7. Procedure ES | Initialize element matrices. |
| 8. Procedure ES | Calculate element intrinsic stiffness matrices. |
| 9. Processor RSEQ | Resequence nodes for minimum total execution time. |
| 10. Processor TOPO | Form maps to guide assembly and factorization of system matrices. |
| 11. Processor K | Assemble system stiffness matrix. |
| 12. Processor INV | Factor system stiffness matrix. |
| 13. Continue | |
| 14. Processor DRF | Form damage resultant force vectors. |
| 15. Processor SSOL | Solve for static displacements |
| 16. Procedure STRESS | Calculate element stress resultants |
| 17. Processor DGI | Calculate ply level stresses and damage evolution |
| 18. For next load cycle, go to step 13; else stop. | |

B7.2. Progressive Damage Analysis Input

The following listing illustrates the input from a progressive failure analysis. The problem being solved is the uniaxially tensile loaded tapered laminated plate, figure 2, described in the main body of this report. The listing contains the main runstream plus a procedure file to perform the calculations for each load cycle.

```
#
#@\$-me
#
cp $CSM_PRC/proclib.gal proclib.gal      .Copy procedure library
chmod u+w proclib.gal
testbed << \endinput
```

```

*set echo=off                      .Do not echo input
*set plib=28
*open 28 proclib.gal /old          .Open procedure library
*open/new 1 qoutput.101            .Open output library

.
.   tapered panel
.   EX47 4 node quad elements
.   24 nodes, 14 elements
.
*add pffc.clp                      .Add procedure for repeating calculations
*def/a es_name = EX47              .Element name
*def/a es_proc = ES1              .Element processor name
*call ES ( function='DEFINE ELEMENTS'; es_proc = <es_proc> ;--
      es_name=<es_name> )
.
[xqt TAB
  START 24                        .24 nodes
  JOINT LOCATIONS                 .Enter joint locations
  1  0.0 0.0 0.0 20.0 2.0 0.0 8 1 3
  8  0.0 10.0 0.0 20.0 8.0 0.0
  CONSTRAINT DEFINITION 1        .Constraints:
  zero 1,2,3,4,5: 1,17,8        .Fixed end
  zero 6: 1,24                  .Suppress drilling DOF
.
[xqt AUS                          .Create input datasets
  SYSVEC : appl forc            .Applied forces

  I=1 : J=8 : 3750.0
  I=1 : J=16 : 7500.0
  I=1 : J=24 : 3750.0

.
  TABLE(NI=16,NJ=1): OMB DATA 1 1 .Ply level material data
  IM7/5260
  I=1,2,3,4,5
  J=1: 22.162E+6 0.333 1.262E+6 0.754E+6 0.754E+6
  I=6,7,8,9
  J=1: 0.754E+6 1.0E-4 1.0E-4 0.01
  I=10,11,12,13,14,15,16
  J=1: 0.0 0.0 0.0 0.014 0.014 0.014 0.0

.
  TABLE(NI=3,NJ=3,ITYPE=0): LAM OMB 1 1 .Section properties
  J=1: 1 0.006 0.0
  J=2: 1 0.012 90.0
  J=3: 1 0.006 0.0

.
  TABLE(NI=3,NJ=1,ITYPE=0): DGP DATA 1 1 .Damage evolution data
  J=1: 1.1695 5.5109 3.8686E-7
.
[xqt LAU                          .Create constitutive matrix
  ONLINE=2
[xqt ELD                          .Define connectivity
  <es_expe_cmd>
  NSECT = 1 : SREF=1
    1    2    10    9
    2    3    11    10
    3    4    12    11
    4    5    13    12
    5    6    14    13
    6    7    15    14
    7    8    16    15
    9    10   18    17
   10    11   19    18
   11    12   20    19
   12    13   21    20
   13    14   22    21
   14    15   23    22

```

```

.
[xqt E                      .Initialize element datasets
stop
*call ES (function='INITIALIZE') .initialize element matrices
*call ES (function='FORM STIFFNESS/MATL') .Form stiffness matrices
[xqt RSEQ                   .Resequenece
reset maxcon=12
[xqt TOPO                   .Create maps
[xqt K                      .Assemble global stiffness matrix
[xqt INV                    .Factor the global stiffness matrix
.
*def/i ns_overwrite=<true>
.
. Call procedure to perform calculations at each cycle
.
*call PFFC ( es_proc=<es_proc> ; es_name=<es_name> ; --
N_fcycl=1 ; N_lcycl=2000 ; N_cylinc=5 ; --
NPRT=100 )
.
*pack 1
[xqt exit
\endinput

B7.2.1. Procedure to perform the loop through the calculations for each load cycle
(file name pffc.clp)

*procedure PFFC ( es_proc ; es_name ; --
N_fcycl ; N_lcycl ; N_cylinc ; --
NPRT )
.
. N_fcycl: first fatigue cycle
. N_lcycl: last fatigue cycle
. N_cylinc: cycle increment
. NPRT: output storage cycle increment
.
. begin loop here
.
*set echo=off
*def icount = 0 .Initialize print counter
*DO :CYCLOOP $NCYL = <[N_fcycl]>, <[N_lcycl]>, <[N_cylinc]>
*def icount = ( <icount> + 1 )
*if < <icount> /eq <[NPRT]> /or <$NCYL> /eq 1 > /then
*def iprint = 1
*def icount = 0
*else
*def iprint = 0
*endif
*def delinc = <[N_cylinc]>
.
[xqt DRF                      .Calculate damage resultant forces
select /PRINT = 0
stop
.
[xqt SSOL                    .Solve for static displacements
.
. Calculate elemental stress resultants
.
*call STRESS (direction=1; location= INTEG_PTS; print=<false> )
.
*if < <IPRINT> /eq 1 > /then
[xqt VPRT                    .Print static displacements
format = 4
print STAT DISP
stop

```

```

      [xqt DGI          .Calculate ply level stresses,
      select /PRINT = 2          .strains, and damage evolution
      select /INC_SIZE = <delinc>
      select /N_CYCLE = <$NCYL>
      select /NINCR = 1
      select /NINCR = <$SNCYL>
      stop

*endif

*if < <IPRINT> /ne 1 > /then
[xqt DGI          .Calculate ply level stresses,
      select /PRINT = 0          .strains, and damage evolution
      select /INC_SIZE = <delinc>
      select /N_CYCLE = <$NCYL>
      select /NINCR = 1
      select /NINCR = <$SNCYL>
      stop

*endif

*if < <IPRINT> /eq 1 > /then          .Store datasets for post processing

*copy 1, PLYDT.<ES_NAME>.<$NCYL>.1 = 1, PDAT.<ES_NAME>.*
*copy 1, DISP.<ES_NAME>.<$NCYL>.1 = 1, STAT.DISP.*

*endif

:CYCLOOP

*set echo=off
*end

```

B7.3. Progressive Damage Analysis Output

The following is a partial list of a progressive failure analysis output produced by processor DGI. Data for postprocessing is stored in dataset PLYDT.xxxx.xxx.1 .

```

** BEGIN DGI ** DATA SPACE= 2000000 WORDS
CYCLE NUM. = 496

```

```

ELEMENT NUMBER 1 TYPE EX47
EVALUATION (INTG) POINT NUMBER 1
REFERENCE SURFACE STRAINS AND CURVATURES
      E0-X      E0-Y      E0-XY      K-X      K-Y      K-XY
0.4619E-02 -0.6946E-04 0.1180E-02 0.0000E+00 0.0000E+00 0.0000E+00
COMBINED BENDING AND MEMBRANE STRESSES, STRAINS, AND MATRIX CRACK DAMAGE
VARIABLE
FOR EACH LAYER OF ELEMENT 1 TYPE EX47

```

| LAYER | THETA | SIG-1 | SIG-2 | TAU-12 | STRAIN-1 | STRAIN-2 | GAMMA-12 |
|-------|-------|-----------|-----------|------------|------------|------------|------------|
| 1 | 0. | 0.103E+06 | 0.187E+04 | 0.890E+03 | 0.462E-02 | -0.695E-04 | 0.118E-02 |
| 2 | 90. | 0.384E+03 | 0.578E+04 | -0.890E+03 | -0.695E-04 | 0.462E-02 | -0.118E-02 |
| 3 | 0. | 0.103E+06 | 0.187E+04 | 0.890E+03 | 0.462E-02 | -0.695E-04 | 0.118E-02 |

| LAYER | ALPM-11 | ALPM-22 | ALPM-12 |
|-------|-----------|-----------|-----------|
| 1 | 0.000E+00 | 0.122E-11 | 0.000E+00 |
| 2 | 0.000E+00 | 0.473E-04 | 0.000E+00 |
| 3 | 0.000E+00 | 0.122E-11 | 0.000E+00 |

```

** BEGIN DGI ** DATA SPACE= 2000000 WORDS
CYCLE NUM. = 996

```

ELEMENT NUMBER 1 TYPE EX47

EVALUATION (INTG) POINT NUMBER 1

REFERENCE SURFACE STRAINS AND CURVATURES

| E0-X | E0-Y | E0-XY | K-X | K-Y | K-XY |
|------------|-------------|------------|------------|------------|------------|
| 0.4623E-02 | -0.6882E-04 | 0.1183E-02 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |

COMBINED BENDING AND MEMBRANE STRESSES, STRAINS, AND MATRIX CRACK DAMAGE

VARIABLE

FOR EACH LAYER OF ELEMENT 1 TYPE EX47

| LAYER | THETA | SIG-1 | SIG-2 | TAU-12 | STRAIN-1 | STRAIN-2 | GAMMA-12 | |
|-------|-------|-----------|-----------|------------|------------|------------|------------|--|
| 1 | 0. | 0.103E+06 | 0.187E+04 | 0.892E+03 | 0.462E-02 | -0.688E-04 | 0.118E-02 | |
| 2 | 90. | 0.382E+03 | 0.573E+04 | -0.892E+03 | -0.688E-04 | 0.462E-02 | -0.118E-02 | |
| 3 | 0. | 0.103E+06 | 0.187E+04 | 0.892E+03 | 0.462E-02 | -0.688E-04 | 0.118E-02 | |

| LAYER | ALPM-11 | ALPM-22 | ALPM-12 |
|-------|-----------|-----------|-----------|
| 1 | 0.000E+00 | 0.246E-11 | 0.000E+00 |
| 2 | 0.000E+00 | 0.901E-04 | 0.000E+00 |
| 3 | 0.000E+00 | 0.246E-11 | 0.000E+00 |

** BEGIN DGI ** DATA SPACE= 2000000 WORDS

CYCLE NUM. = 1496

ELEMENT NUMBER 1 TYPE EX47

EVALUATION (INTG) POINT NUMBER 1

REFERENCE SURFACE STRAINS AND CURVATURES

| E0-X | E0-Y | E0-XY | K-X | K-Y | K-XY |
|------------|-------------|------------|------------|------------|------------|
| 0.4625E-02 | -0.6839E-04 | 0.1184E-02 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |

COMBINED BENDING AND MEMBRANE STRESSES, STRAINS, AND MATRIX CRACK DAMAGE

VARIABLE

FOR EACH LAYER OF ELEMENT 1 TYPE EX47

| LAYER | THETA | SIG-1 | SIG-2 | TAU-12 | STRAIN-1 | STRAIN-2 | GAMMA-12 | |
|-------|-------|-----------|-----------|------------|------------|------------|------------|--|
| 1 | 0. | 0.103E+06 | 0.187E+04 | 0.893E+03 | 0.463E-02 | -0.684E-04 | 0.118E-02 | |
| 2 | 90. | 0.376E+03 | 0.568E+04 | -0.893E+03 | -0.684E-04 | 0.463E-02 | -0.118E-02 | |
| 3 | 0. | 0.103E+06 | 0.187E+04 | 0.893E+03 | 0.463E-02 | -0.684E-04 | 0.118E-02 | |

| LAYER | ALPM-11 | ALPM-22 | ALPM-12 |
|-------|-----------|-----------|-----------|
| 1 | 0.000E+00 | 0.372E-11 | 0.000E+00 |
| 2 | 0.000E+00 | 0.129E-03 | 0.000E+00 |
| 3 | 0.000E+00 | 0.372E-11 | 0.000E+00 |

** BEGIN DGI ** DATA SPACE= 2000000 WORDS

CYCLE NUM. = 1996

ELEMENT NUMBER 1 TYPE EX47

EVALUATION (INTG) POINT NUMBER 1

REFERENCE SURFACE STRAINS AND CURVATURES

| E0-X | E0-Y | E0-XY | K-X | K-Y | K-XY |
|------------|-------------|------------|------------|------------|------------|
| 0.4627E-02 | -0.6806E-04 | 0.1185E-02 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |

COMBINED BENDING AND MEMBRANE STRESSES, STRAINS, AND MATRIX CRACK DAMAGE

VARIABLE

FOR EACH LAYER OF ELEMENT 1 TYPE EX47

| LAYER | THETA | SIG-1 | SIG-2 | TAU-12 | STRAIN-1 | STRAIN-2 | GAMMA-12 | |
|-------|-------|-----------|-----------|------------|------------|------------|------------|--|
| 1 | 0. | 0.103E+06 | 0.187E+04 | 0.894E+03 | 0.463E-02 | -0.681E-04 | 0.119E-02 | |
| 2 | 90. | 0.370E+03 | 0.564E+04 | -0.894E+03 | -0.681E-04 | 0.463E-02 | -0.119E-02 | |
| 3 | 0. | 0.103E+06 | 0.187E+04 | 0.894E+03 | 0.463E-02 | -0.681E-04 | 0.119E-02 | |

| LAYER | ALPM-11 | ALPM-22 | ALPM-12 |
|-------|-----------|-----------|-----------|
| 1 | 0.000E+00 | 0.500E-11 | 0.000E+00 |
| 2 | 0.000E+00 | 0.164E-03 | 0.000E+00 |
| 3 | 0.000E+00 | 0.500E-11 | 0.000E+00 |

B8. Structure of Datasets Unique to Processor DGI

B8.1. PDAT.xxxx

Data set PDAT.xxxx contains ply-level damage dependent stresses, strains, and internal state variables. Data are centroidal values. The variable xxxx is the element name. The data for each element is stored in a record named DAT_PLY.ielt, where ielt is the element number. Each record contains n items, where

$$n = nlayer \times 9$$

and $nlayer$ is the number of layers in the model.

The data is expressed with respect to ply coordinates and is stored in the following order:

1. σ_{11} normal stress in the fiber direction.
2. σ_{22} normal stress transverse to the fibers.
3. σ_{12} shear stress.
4. ϵ_{11} strain in the fiber direction.
5. ϵ_{22} strain transverse to the fibers.
6. ϵ_{12} shearing strain.
7. α_{L11}^M internal state variable associated with fiber fracture.
8. α_{L22}^M internal state variable associated with mode I opening of the matrix crack.
9. α_{12}^L internal state variable associated with mode II opening of the matrix crack.

Repeated $nlayer$ times.

B8.2. DGP.DATA.1.1

This data set is created by AUS/TABLE and contains the growth law parameters for the matrix cracking evolutionary relationship. The following variables are used to specify table size:

NI = number of material parameters, for this case 3

NJ = number of material systems, for this case 1

Type = numerical format such as real or integer

where NI and NJ are the number of columns and rows, respectively and Type specifies numerical format, real or integer.

Each entry contains the following:

1. Growth law parameter, \tilde{k} .
2. Growth law parameter, \tilde{n} .
3. Parameter for determining $\frac{d\alpha_{Lij}}{dS}$, $dpara$

These entries are repeated NJ times.

B8.3. ISV.xxxx

This data set contains the matrix cracking internal state variables at each layer. The variable xxxx is the element name. The data is stored in a record named ALPAM.1.

This record contains n items, where

$$n = nlayer \times nintgpt \times nelt$$

and $nlayer$ is the number of layers in the model, $nintgpt$ is the number of integration points for the element, and $nelt$ is the number of elements.

The data is stored in the following order:

1. α_{L11}^M , internal state variable associated with fiber fracture.
2. α_{L22}^M , internal state variable associated with mode I opening of the matrix crack.
3. α_{L12}^M , internal state variable associated with mode II opening of the matrix crack.

The data storage occurs for every layer, every integration point, and every element.

Appendix C

Residual Strength Runstream

C1. General Description

This appendix lists a sample runstream that was used to calculate the residual strength of an orthotropic center-notched laminate that was monotonically loaded to failure.

C2. Residual Strength Analysis Input

The following listing illustrates the input from a residual strength analysis. The problem being solved is the uniaxially tensile loaded center-notched orthotropic laminated plate shown in figure 8. The listing contains the main runstream plus a procedure file. The procedure file calculates the response during the monotonic loading to failure and is presented in this appendix. The finite element mesh was created using PATRAN. The file PT2T.PRC was created using the testbed PATRAN To Testbed (PT2T) neutral file converter. This file contains all of the nodal locations, connectivity matrix, boundary conditions, and applied forces.

```
cp $SCSM_PRC/proclib.gal proclib.gal
chmod u+w proclib.gal
testbed > cct.o << \endinput
*set echo=off
*set plib=28
*open 28 proclib.gal /old
*open/new 1 cct.l01
.
.   rectangular panel with center-cut slit
.   quarter panel mesh
.   EX47 4 node quad elements
.
.
*add pffdm.clp
*add initialize.clp
  *ADD PT2T.PRC
*def/a es_name = EX47
*def/a es_proc = ES1
*call ES ( function = 'DEFINE ELEMENTS'; es_proc = <es_proc> ;--
          es_name=<es_name> )
[xqt TAB
  START 800
  *call PT2T_JLOC
.
CONSTRAINT DEFINITION 1
  *call PT2T_BC
.
[xqt AUS
  SYSVEC : appl forc
.
  *call PT2T_AF
.
TABLE(NI=16,NJ=1): OMB DATA 1 1
  AS4/938
  I=1,2,3,4,5
  J=1: 19.60E+6 0.32 1.36E+6 0.72E+6 0.72E+6
  I=6,7,8,9
  J=1: 0.72E+6 1.0E-4 1.0E-4 0.01
  I=10,11,12,13,14,15,16
  J=1: 0.0 0.0 0.0 0.0148 0.005 0.010 0.0
.
TABLE(NI=3,NJ=13,itype=0): LAM OMB 1 1
  J=1: 1 0.0072 -45.0
```

```

J=2: 1 0.0072 45.0
J=3: 1 0.0072 0.0
J=4: 1 0.0072 90.0
J=5: 1 0.0072 -30.0
J=6: 1 0.0072 30.0
J=7: 1 0.0072 0.0
J=8: 1 0.0072 30.0
J=9: 1 0.0072 -30.0
J=10: 1 0.0072 90.0
J=11: 1 0.0072 0.0
J=12: 1 0.0072 45.0
J=13: 1 0.0072 -45.0
.
[xqt LAU
  ONLINE=2
[xqt ELD
  *call PT2T_CONN
.
*def/a solver_name=INV
*call INITIALIZE ( renumber=1; rseq_method=-1; auto_dof_sup=0 )
*call STIFFNESS ( type=MATL )
*print 1 EX47.EFIL.* DATA.1 /m=100
*call FACTOR ( input_matrix=K; output_matrix=K )
*def/i ns_overwrite=<true>
.
*call PFFDM ( es_proc=<es_proc> ; es_name=<es_name> ; --
  N_fcycl=1 ; N_lcycl=1000 ; N_cylinc=1 ;--
  NSUB=0 ; NSTRT=0 ; NS_lcycl=0 ; --
  NPRT=1)
.
*pack 1
[xqt exit
\endinput

```

C2.1 Procedure to perform the monotonic loading calculations (file name pffdm.clp)

```

*procedure PFFDM ( es_proc ; es_name ; --
  N_fcycl ; N_lcycl ; N_cylinc ; --
  NSUB ; NSTRT ; NS_lcycl ; NPRT )
.
  File to control monotonic loading to failure
.
  N_fcycl: first cycle number
.
  N_lcycl: last cycle number
.
  N_cylinc: cycle increment
.
  NSUB: subincrement flag (=0, to bypass)
.
  NSTRT: cycle to start subincrements(=0, to bypass)
.
  NS_lcycl: number of subincrements(=1, to bypass)
.
  NPRT: output storage cycle increment
.
  begin loop here
.
*set echo=off
*def icount = 0
*DO :CYCLOOP $NCRYL = <[N_fcycl]>, <[N_lcycl]>, <[N_cylinc]>
  *def icount = ( <icount> + 1 )
  *if < <icount> /eq <[NPRT]> > /then
    *def iprint = 1
    *def icount = 0
  *else
    *def iprint = 0
  *endif
  *def/i $ITCYCL = 1
  *def/i DUMIT = 1
  *def/i $SNCYL = 1

```

```

*def delinc = <[N_cylinc]>
*def/d st_disp == STAT.DISP.*
*def/d ap_forc == APPL.FORC.*
*def/a nst_disp == NST.DISP
*def/a ost_disp == OST.DISP
*def/a dst_disp == DST.DISP
*DO $ITCYCL = 1,1000
*def/i drfit = 1
[xqt DRF.MNR
  select /PRINT = 0
  select /DSTATUS = 2222
  select /XFACTOR = 1.0
  select /DUMIT = <$ITCYCL>
stop

.
*****
. postpone iterations until damage occurs
.
*G2M /name=drfit /type=D /maxn=1--
  1, DRFITU.EX47.*, DRFDAT.1
*if < <drfit> /eq 0.0 > /then
*def/i $ITCYCL = 1000
*endif
*****
*****
. iterate until change in damage resultant
. forces are small
.
*G2M /name=drfitn /type=I /maxn=1--
  1, DRFITV.EX47.*, DRFDAT.1
*print 1 DRFITV.EX47.* DRFDAT.1 /m=1
*if < <$ITCYCL> /ge 2 > /then
*if < <drfitn> /eq 0 > /then
*def/i $ITCYCL = 1000
*endif
*endif
*****
*if < <$ITCYCL> /ge 2 > /then
  *G2M /name=ddisp /type=D /maxn=1--
    1, DST.DISP, DATA.1 /m=49
  *endif

  *if < <$NCYL> /ge 2 > /then
  [xqt vec
    COMBINE <ost_disp> <- <st_disp>
  *if < <$NCYL> /eq 2 > /then
    COMBINE <dst_disp> <- <st_disp>
  *endif
  *endif

  [xqt SSOL

  *if < <drfit> /ne 0.0 > /then
  *if < <$ITCYCL> /ge 2 > /then
  [xqt vec
    COMBINE <dst_disp> <- <st_disp>
    COMBINE <nst_disp> <- <dst_disp> + <ost_disp>
    COMBINE <st_disp> <- <nst_disp>
  *endif
  *endif

  *call STRESS (direction=1; location= INTEG_PTS; print=<false> )
[xqt DGI
  select /PRINT = 0
  select /INC_SIZE = <delinc>
  select /N_CYCLE = <$NCYL>

```

```

select /NINCR = <$NCRYL>
select /DSTATUS = 22222
stop

*if < <$ITCYCL> /eq 1000 > /then
*if < <IPRINT> /eq 1 > /then
*if < <$NCRYL> /ge 100 > /then

*copy 1, PLYDTM.<ES_NAME>.<$NCRYL>.<$ITCYCL> = 1, PDAT.<ES_NAME>.*
*copy 1, DISPM.<ES_NAME>.<$NCRYL>.<$ITCYCL> = 1, STAT.DISP.*
*endif
*endif
*endif

*ENDDO

CYCLOOP

*set echo=off
*end

```

C3. Residual Strength Analysis Output

Centroidal values of the ply stresses, strains, and internal state variables for each element can be found in the PDAT.xxxx data set (renamed as PLYDTM.EX47.xxxx.yyyy). Nodal displacements are located in the STAT.DISP.xxxx data set (renamed as DISPM.EX47.xxxx.yyyy). These data sets are located in the *.l01 output file and can be retrieved as separate individual files. An example of data output for one damaged element from one of the PDAT.xxxx data sets is given below.

Record DAT_PLY.97 of dataset PLYDTM.EX47.190.50

```

1: 1.9230D+05 6.8000D+02 7.2000D+03 9.8001D-03 1.0500D-02 3.1344D-02
7: 0.0000D+00 1.3139D-02 2.1344D-02 2.0601D+05 6.8000D+02 7.2000D+03
13: 1.0500D-02 9.8001D-03 -3.1344D-02 0.0000D+00 1.2664D-02 -4.1344D-02
19: 2.9008D+04 -6.8656D+03 5.0370D+02 2.5822D-02 -5.5218D-03 6.9959D-04
25: 2.4230D-02 0.0000D+00 0.0000D+00 -1.0801D+05 6.8000D+02 -5.0370D+02
31: -5.5218D-03 2.5822D-02 -6.9958D-04 0.0000D+00 2.3558D-02 0.0000D+00
37: 1.4610D+05 5.0053D+03 7.2000D+03 1.7683D-02 2.6170D-03 2.7494D-02
43: 1.0311D-02 1.3219D-03 1.7494D-02 1.5110D+05 1.8871D+03 7.2000D+03
49: 1.8289D-02 2.0111D-03 -2.6794D-02 1.0610D-02 3.0906D-03 -3.6794D-02
55: 2.9008D+04 -6.8656D+03 5.0370D+02 2.5822D-02 -5.5218D-03 6.9959D-04
61: 2.4230D-02 0.0000D+00 0.0000D+00 1.5110D+05 1.8871D+03 7.2000D+03
67: 1.8289D-02 2.0111D-03 -2.6794D-02 1.0610D-02 3.0906D-03 -3.6794D-02
73: 1.4610D+05 5.0053D+03 7.2000D+03 1.7683D-02 2.6170D-03 2.7494D-02
79: 1.0311D-02 1.3219D-03 1.7494D-02 -1.0801D+05 6.8000D+02 -5.0370D+02
85: -5.5218D-03 2.5822D-02 -6.9958D-04 0.0000D+00 2.3558D-02 0.0000D+00
91: 2.9008D+04 -6.8656D+03 5.0370D+02 2.5822D-02 -5.5218D-03 6.9959D-04
97: 2.4230D-02 0.0000D+00 0.0000D+00 2.0601D+05 6.8000D+02 7.2000D+03
103: 1.0500D-02 9.8001D-03 -3.1344D-02 0.0000D+00 1.2664D-02 -4.1344D-02
109: 1.9230D+05 6.8000D+02 7.2000D+03 9.8001D-03 1.0500D-02 3.1344D-02
115: 0.0000D+00 1.3139D-02 2.1344D-02

```

This data is for element number 97 at load step number 190 and at the 50th (or last) iteration. Based on the structure of data sets in Appendix B Section 8, the underlined data is the output for the third ply (in this case that would be the 0° ply). The number 2.9008D+04 means that σ_{11} , the normal stress in the fiber direction, is 29,008 psi. This is 10% of the failure strength because we know that fiber fracture has occurred from the value of α_{11} , 2.4230D-02.

An example of the nodal displacements for the first nine nodes are listed below. Each row of data belongs to one node. The order of data is such that the x-direction translation is listed first, then the y and z translations followed by the rotations about the x, y, and z axes, respectively.

Record DATA.1 of dataset DISPM.EX47.190.50

```

1: 1.9339E-02 3.7506E-03 0.0000E+00 0.0000E+00 0.0000E+00 0.0000E+00

```

| | | | | | | |
|-----|------------|------------|------------|------------|------------|------------|
| 7: | 1.9377E-02 | 3.2392E-03 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |
| 13: | 1.9422E-02 | 2.7315E-03 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |
| 19: | 1.9666E-02 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |
| 25: | 1.9475E-02 | 2.2355E-03 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |
| 31: | 1.9656E-02 | 4.2360E-04 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |
| 37: | 1.9532E-02 | 1.7568E-03 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |
| 43: | 1.9628E-02 | 8.5391E-04 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |
| 49: | 1.9585E-02 | 1.2969E-03 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 | 0.0000E+00 |

How often such data is stored in data sets is up to the user and is controlled by the *NPRT* variable in the *runstream* and the **copy 1* command in the procedure *pffdm.clp*.

References

1. Stewart, C.B.; "The Computational Structural Mechanics Testbed User's Manual," *NASA Technical Memorandum 100644*, October 1989.
2. Allen, D.H., Harris, C.E., and Groves, S.E.; "A Thermomechanical Constitutive Theory for Elastic Composites with Distributed Damage--Part I: Theoretical Development," *International Journal of Solids and Structures*, Vol. 23, No. 9, 1987, pp.1301-1318.
3. Allen, D.H., Harris, C.E., and Groves, S.E.; "A Thermomechanical Constitutive Theory for Elastic Composites with Distributed Damage--Part II: Application to Matrix Cracking in Laminated Composites," *International Journal of Solids and Structures*, Vol. 23, No. 9, 1987, pp.1319-1338.
4. Vakulenko, A.A. and Kachanov, M.L.; "Continuum Theory of a Medium with Cracks," *Mekhanika Tverdogo Tela*, Vol. 6, No. 4, 1971, pp. 159-166.
5. Lee, J.W., Allen, D.H., and Harris, C.E.; "Internal State Variable Approach for Predicting Stiffness Reductions in Fibrous Laminated Composites with Matrix Cracks," *Journal of Composite Materials*, Vol. 23, 1989, pp.1273-1291.
6. Jones, R.M.; *Mechanics of Composite Materials*, Hemisphere Publishing Corporation, New York, 1975.
7. Allen, D.H., Nottorf, E.W., and Harris, C.E.; "Effect of Microstructural Damage on Ply Stresses in Laminated Composites," *Recent Advances in the Macro- and Micro-Mechanics of Composite Materials Structures-Proceedings of the Symposium*, ASME, 1988, pp. 135-145.
8. Chou, P.C., Wang, A.S.D., and Miller, H., "Cumulative Damage Model for Advanced Composite Materials," *AFWAL-TR-82-4083*, U.S. Air Force, August 1982. (Also available from DTIC as AD-A122859.)
9. Lo, D.C., Allen, D.H., and Harris, C.E.; "A Continuum Model for Damage Evolution in Laminated Composites," *Inelastic Deformation of Composite Materials*, G.J. Dvorak, ed., Springer -Verlag, 1990, pp. 549-561.
10. Lo, D.C., Allen, D.H., and Harris, C.E.; "A Procedure for Utilization of a Damage-Dependent Constitutive Model for Laminated Composites," *NASA Technical Memorandum 104219*, February 1992.
11. Coats, T.W.; "Experimental Verification of a Progressive Damage Model for Composite Laminates Based on Continuum Damage Mechanics," *NASA Contractor Report 195020*, Dec. 1994.
12. Buie, K.D.; "A Finite Element Model for Laminated Composite Plates with Matrix Cracks and Delaminations," M.S. Thesis, Texas A&M University, December 1988.
13. Tay, T.E., Lam, K.Y., and Cen, Z., "Analysis of Composite Structures with Distributed and Localized Damage by the Finite-Element Method," *Composite Structures*, Vol. 37, 1997, pp. 135-143.
14. Stanley, G.M., "The Computational Structural Mechanics Testbed Structural Element Processor ES1: Basic SRI and ANS Shell Elements," *NASA Contractor Report 4357*, 1991.
15. Coats, T.W. and Harris, C.E., "A Progressive Damage Methodology for Residual Strength Predictions of Notched Composite Panels" *NASA Technical Memorandum 207646*, April, 1998.
16. Scholz, D.B., Dost, E.F., and Flynn, B.W., "Advanced Technology Composite Fuselage - Materials and Processes," *NASA Contractor Report 4731*, 1995.
17. Lagace, P.A., Bhat, N.V., and Gundogdu, A., "Response of Notched Graphite/Epoxy and Graphite/Peek Systems," *Composite Materials: Fatigue and Fracture*, Fourth Volume, ASTM STP 1156, W.W.

Stinchcomb and N.E. Ashbaugh, Eds., American Society for Testing and Materials, Philadelphia, 1993, pp. 55-71.

18. Coquill, S.L. and Adams, D.F., "Mechanical Properties of Several Neat Polymer Matrix Materials and Unidirectional Carbon Fiber-Reinforced Composites," *NASA Contractor Report* 181805, April 1989.

Table 1. Material Properties of Unidirectional Ply of IM7/5260

| | |
|---------------------------|----------------------|
| E_{11} , Msi | 22.16 |
| E_{22} , Msi | 1.26 |
| G_{12} , Msi | 0.75 |
| ν_{12} | 0.333 |
| t_{ply} , in. | 0.006 |
| ϵ_{11crit} | 0.015 |
| ϵ_{22crit} | 0.008 |
| Growth law parameters: | |
| \bar{K} | 1.1695 |
| \bar{n} | 5.5109 |
| d_{para} | 3.8686×10^7 |

Table 2. Maximum Fatigue Loads Employed in Sample Calculations

| Layup | Specimen geometry | Maximum fatigue load (R = 0.1) |
|-----------------------------------|-------------------|--------------------------------|
| [0/± 45/90] _s | unnotched | 3300 lb/in |
| | open hole | 2000 lb/in |
| [0/90 ₃] _s | unnotched | 2480 lb/in |
| | open hole | 1572 lb/in |

Table 3. Lamina Material Properties

| Material | E_{11} (Msi) | E_{22} (Msi) | G_{12} (Msi) | ν_f (%) | ν_{12} | ϵ_{cr}^{90} (%) | ϵ_{cr}^0 (%) | $*\gamma_{12}^0$ (%) | $*\gamma_{12}^{cr}$ (%) |
|--------------------------|-------------------|-------------------|-------------------|----------------|------------|-----------------------------|--------------------------|-------------------------|----------------------------|
| ^a AS4/8553-40 | 19.7 | 1.31 | 0.65 | 58.2 | 0.34 | 0.87 | 1.56 | 1.00 | 10.00 |
| ^a AS4/938 | 19.6 | 1.36 | 0.72 | 57.2 | 0.32 | 0.50 | 1.48 | 1.00 | 10.00 |
| ^b AS4/3501-6 | 20.0 | 1.36 | 0.87 | 60.3 | 0.28 | 0.50 | 1.50 | 1.00 | 10.00 |

a E_{11} , E_{22} , G_{12} , ν_{12} , ϵ_{cr}^{90} , and ϵ_{cr}^0 measured by Boeing (ref. 16)

b E_{11} , E_{22} , G_{12} , ν_{12} , ϵ_{cr}^{90} , and ϵ_{cr}^0 measured by Lagace et al. (ref. 17)

* γ_{12}^0 and γ_{12}^{cr} are approximations from Iosipescu shear test data (ref. 18)

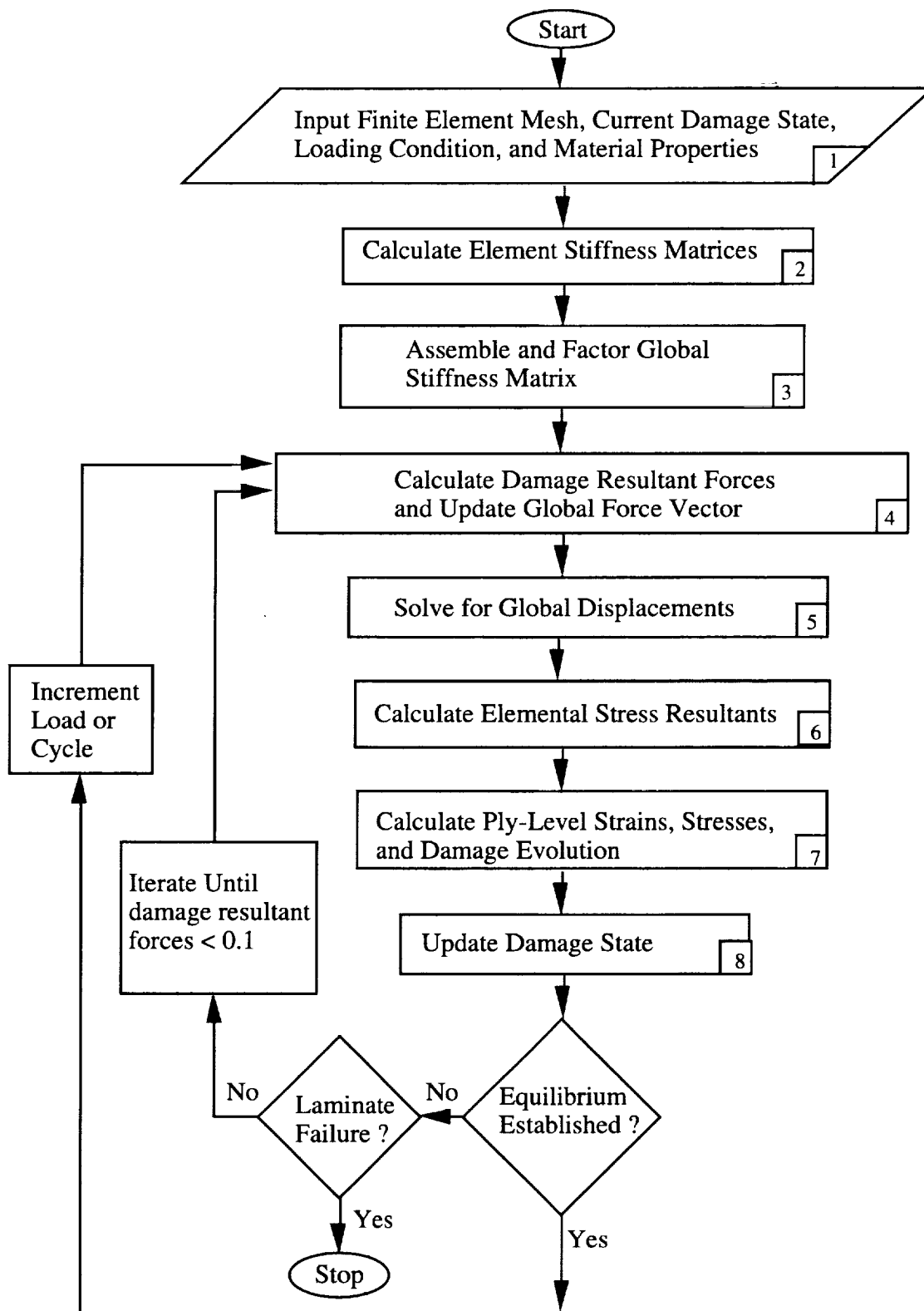
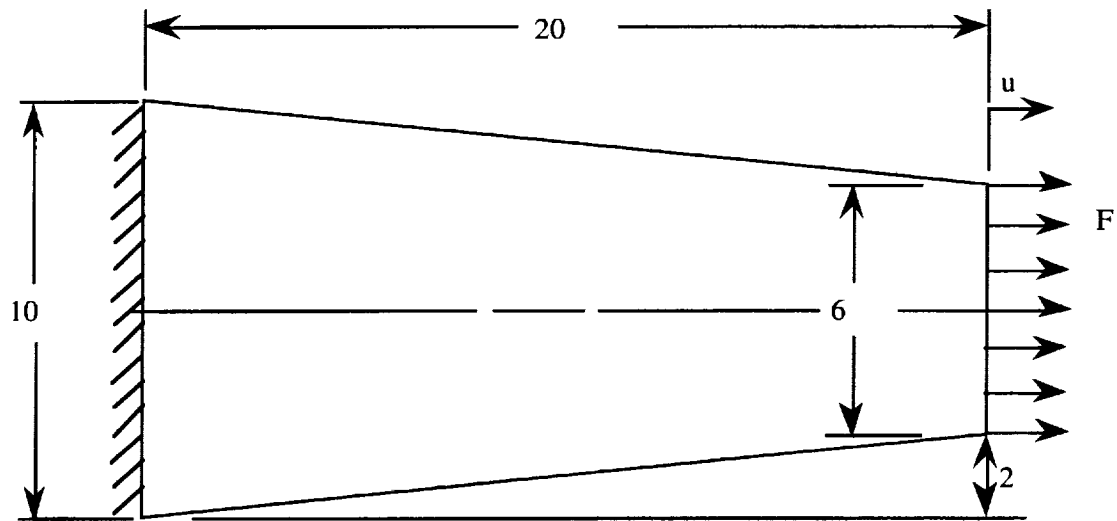
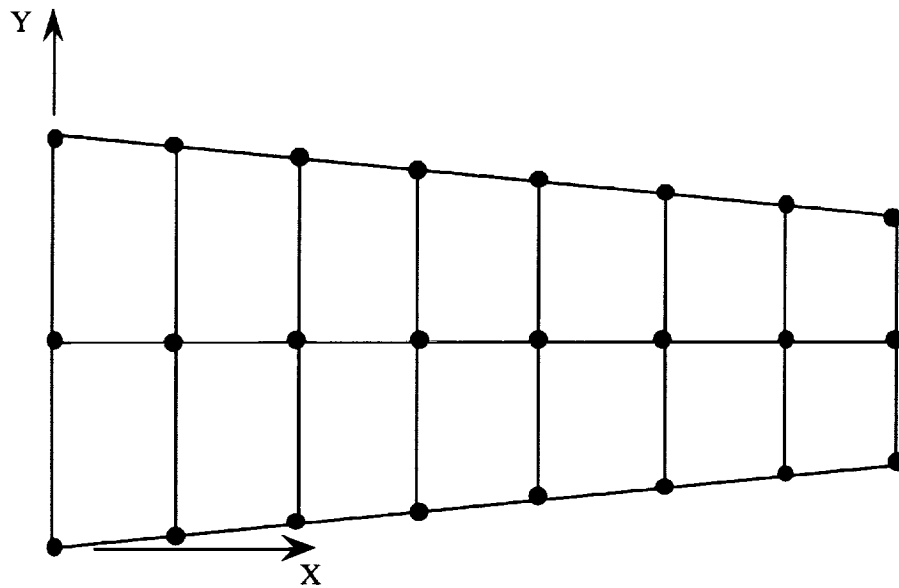


Figure 1. Progressive failure analysis scheme.



(a) Dimensions and boundary conditions.



(b) The finite element model in sample calculation.

Figure 2. Conditions and model of cross-ply laminated composite plate. All linear dimensions are in inches.

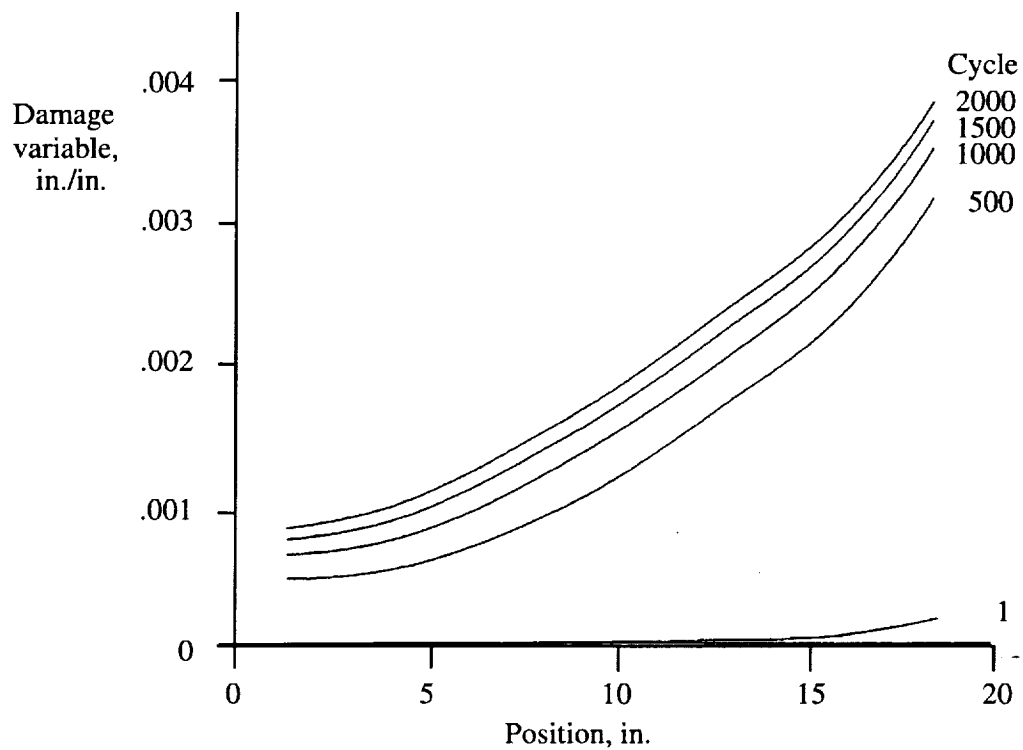


Figure 3. Averaged distribution of mode I matrix crack damage variable α_{22}^M in 90° plies.

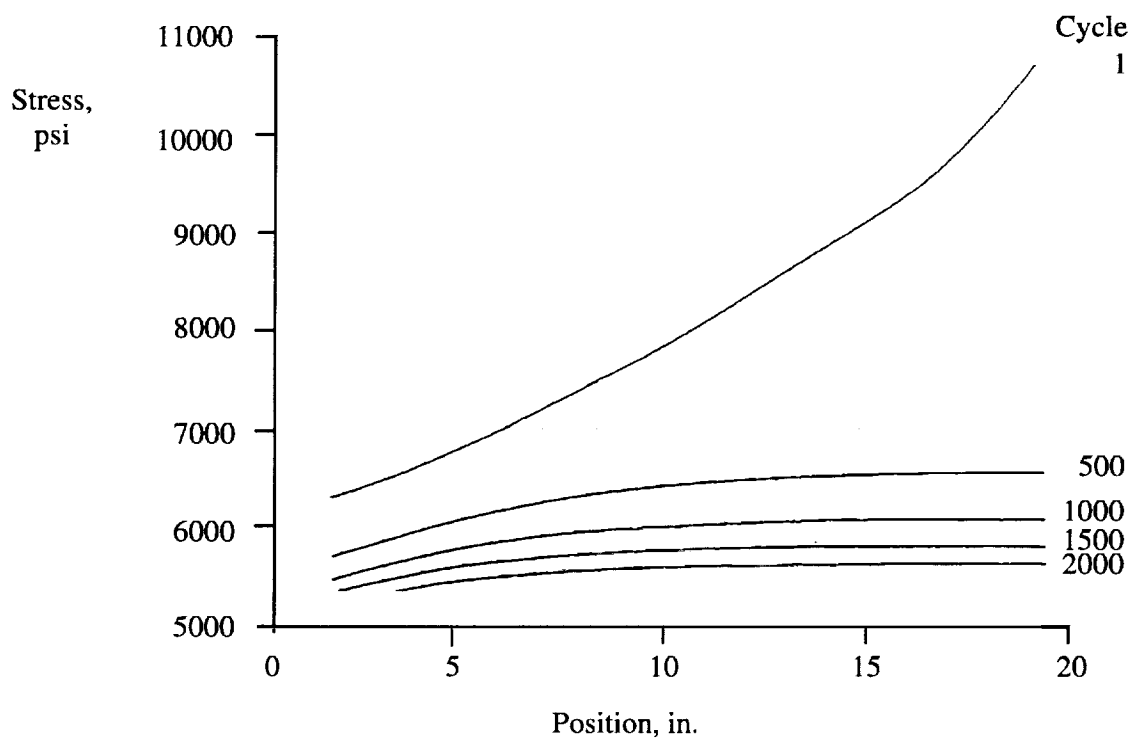


Figure 4. Distribution of stress component normal to fibers in 90° plies.

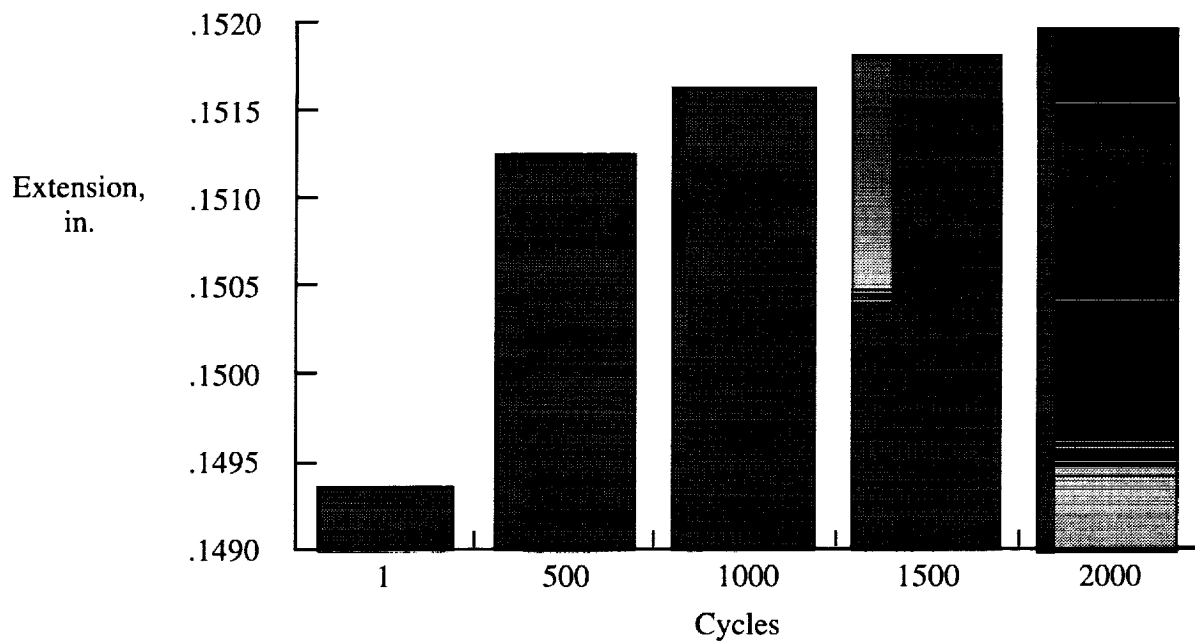


Figure 5. Global displacements resulting from load redistribution.

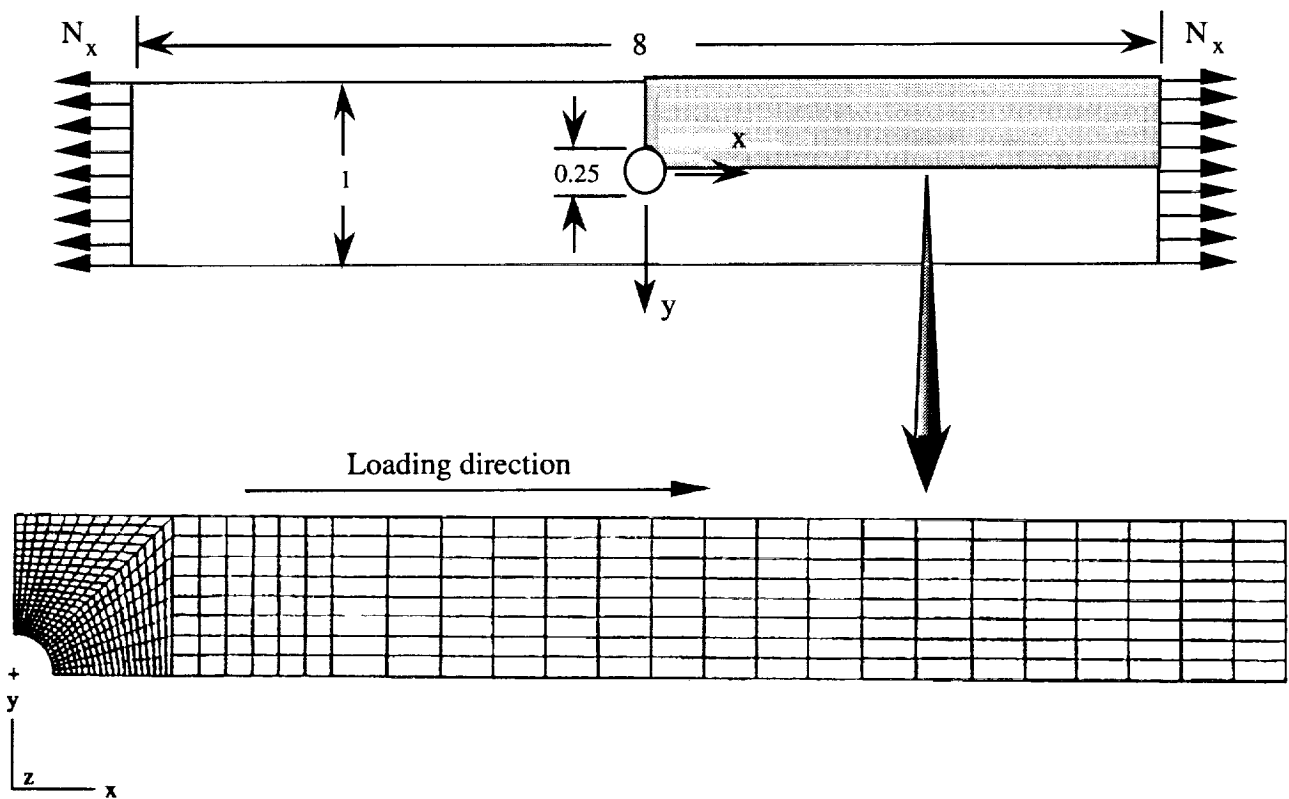


Figure 6. Finite element model for a laminate with a central circular hole.

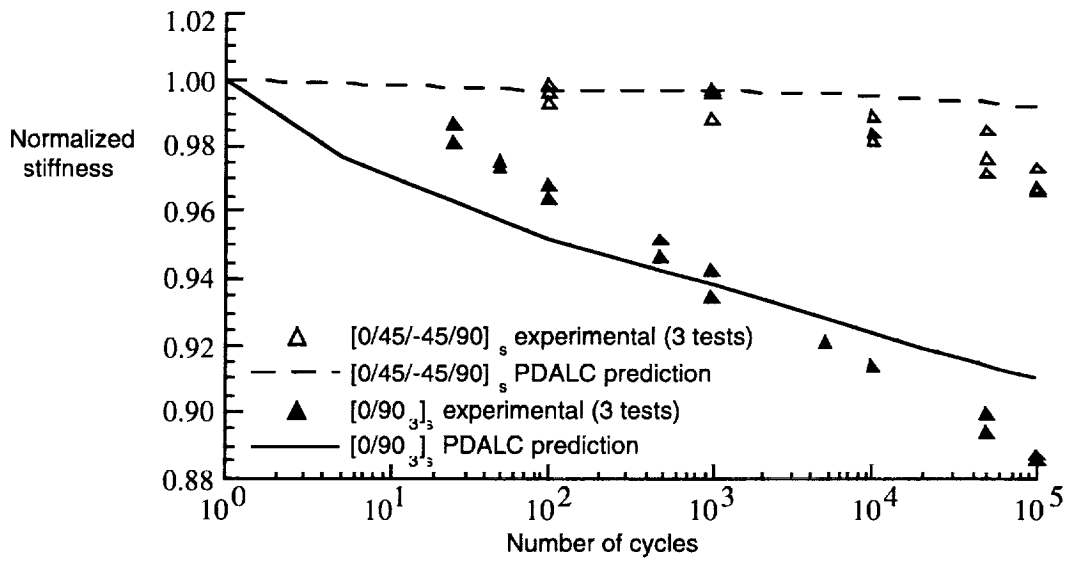


Figure 7. Stiffness loss of IM7/5260 laminates with central circular notch.

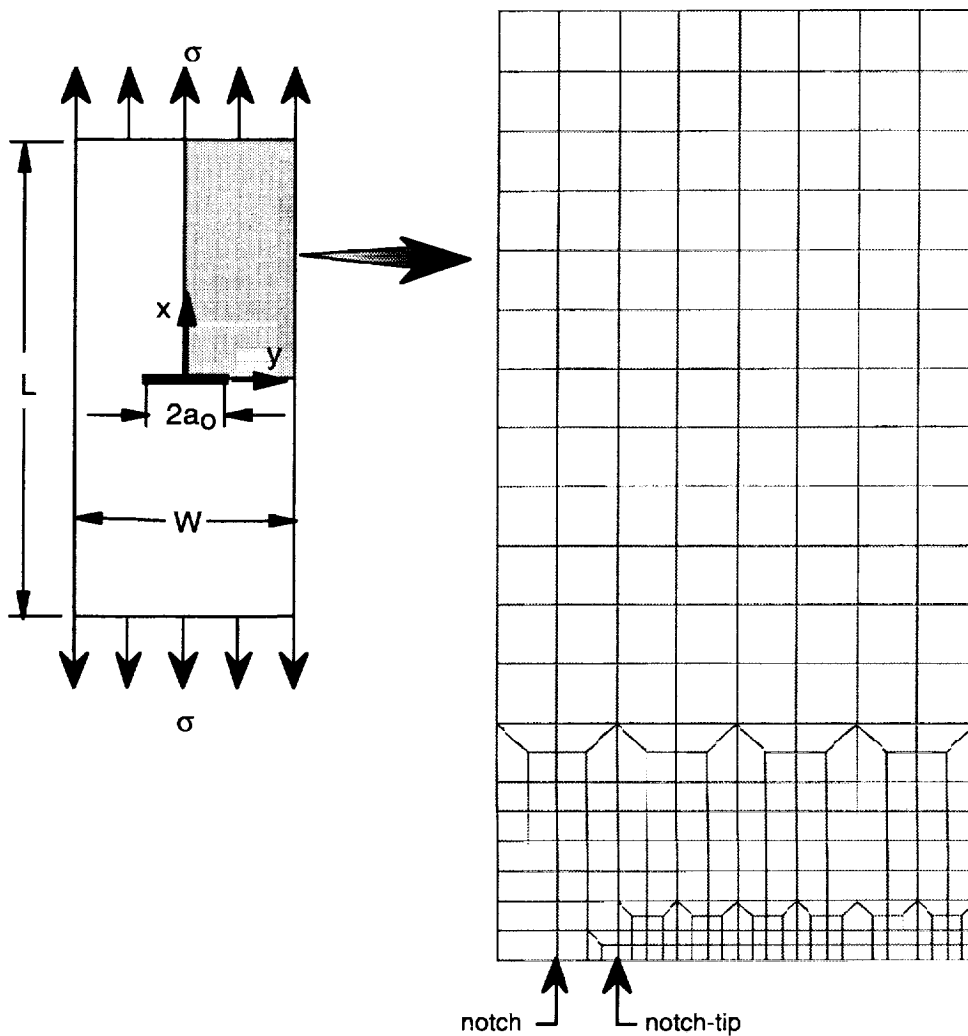


Figure 8. Finite Element Mesh of the Center-Crack Tension Panel.

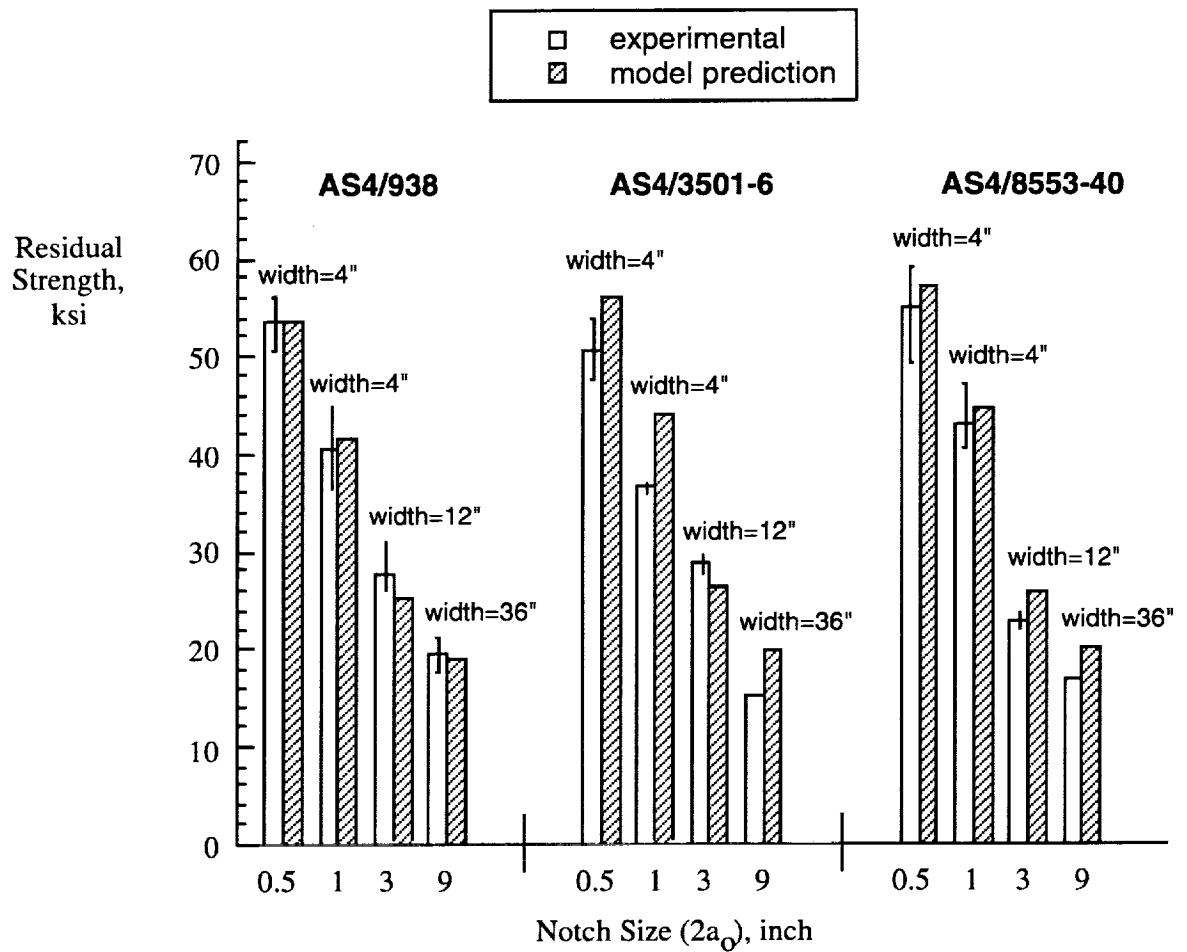


Figure 9. Residual Strength Predictions for the Center-Crack Tension Panels.

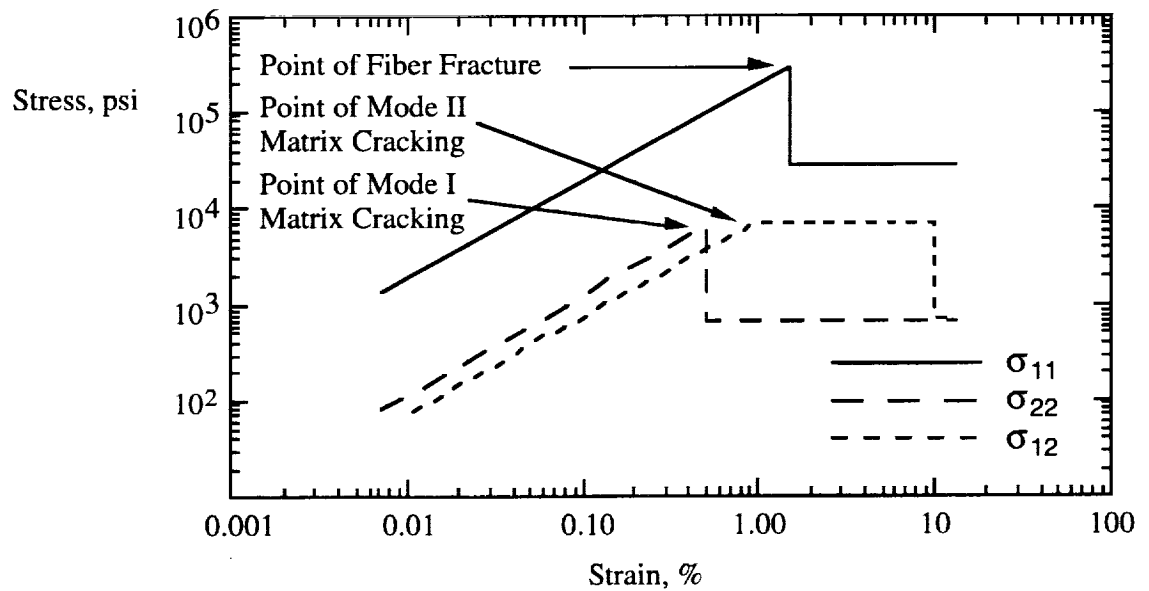


Figure 10. Load Carrying Capability After Ply Failure.

| REPORT DOCUMENTATION PAGE | | | Form Approved OMB No. 0704-0188 | |
|---|---|--|--|--|
| Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503. | | | | |
| 1. AGENCY USE ONLY (Leave blank) | | 2. REPORT DATE October 1998 | | 3. REPORT TYPE AND DATES COVERED Technical Memorandum |
| 4. TITLE AND SUBTITLE Progressive Damage Analysis of Laminated Composite (PDALC) - (A Computational Model Implemented in the NASA COMET Finite Element Code) Version 2.0 | | | 5. FUNDING NUMBERS WU 538-13-11-05 | |
| 6. AUTHOR(S) Timothy W. Coats, Charles E. Harris, David C. Lo, and David H. Allen | | | | |
| 7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Langley Research Center Hampton, VA 23681-2199 | | | 8. PERFORMING ORGANIZATION REPORT NUMBER L-17761 | |
| 9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001 | | | 10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA/TM-1998-208718 | |
| 11. SUPPLEMENTARY NOTES | | | | |
| 12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified-Unlimited Subject Category 24 Distribution: Standard Availability: NASA CASI (301) 621-0390 | | | 12b. DISTRIBUTION CODE | |
| 13. ABSTRACT (Maximum 200 words) A method for analysis of progressive failure in the Computational Structural Mechanics Testbed is presented in this report. The relationship employed in this analysis describes the matrix crack damage and fiber fracture via kinematics-based volume-averaged damage variables. Damage accumulation during monotonic and cyclic loads is predicted by damage evolution laws for tensile load conditions. The implementation of this damage model required the development of two testbed processors. While this report concentrates on the theory and usage of these processors, a complete listing of all testbed processors and inputs that are required for this analysis are included. Sample calculations for laminates subjected to monotonic and cyclic loads were performed to illustrate the damage accumulation, stress redistribution, and changes to the global response that occurs during the loading history. Residual strength predictions made with this information compared favorably with experimental measurements. | | | | |
| 14. SUBJECT TERMS Composites; Damage tolerance; Fracture mechanics; Progressive damage; Residual strength | | | 15. NUMBER OF PAGES 41 | |
| | | | 16. PRICE CODE A03 | |
| 17. SECURITY CLASSIFICATION OF REPORT Unclassified | 18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified | 19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified | 20. LIMITATION OF ABSTRACT | |

